MANUAL OF AVEVA EVERYTHING 3D

Contents

A. Module Model

1. Interface

1.1 Getting to start

To lunch AVEVA E3D select:

➔ **Start > All Program > AVEVE > AVEVA Everything3D 2.10** (from the task bar)

➔ Double click **the AVEVE > AVEVA Everything3D 2.10 (**shortcut on desktop)

The AVEVA E3D initialization image will be display:

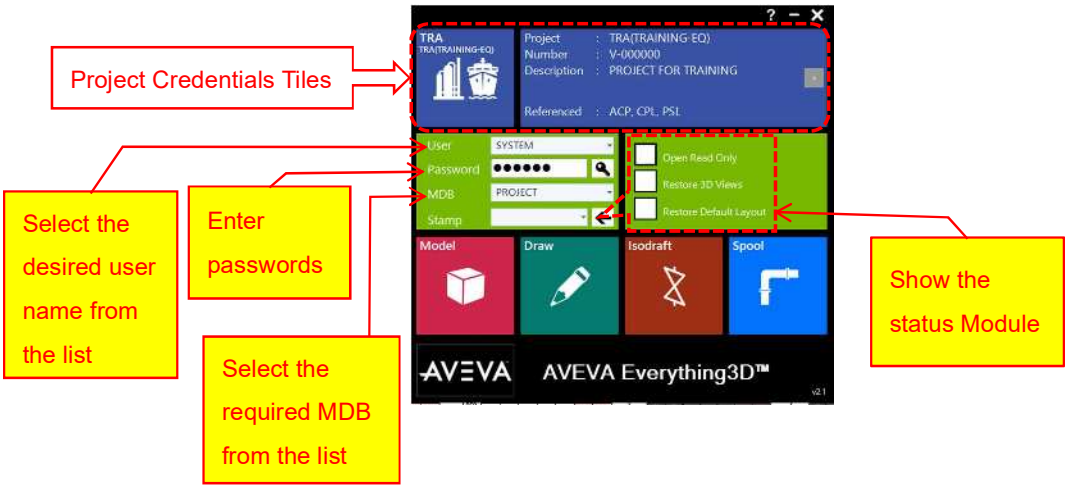
When initialization complete, The AVEVA E3D Login form will be displayed:

➔ **Project selection Tiles**: field list all available project that allow the user to select a project to work on.

➔ **Project search Tile:** allows the user to search for a project and to display masked project tiles.

➔ **Module selection Tiles**: starts the selected module with the selected project and credentials.

The Project Information Tile will be show when user click TRA project:



The access rights will be applied on databases loaded by the MDB depending on the User selected in the Project Credential tiles.

EX: The user would like to make 3D model equipment of TRA project

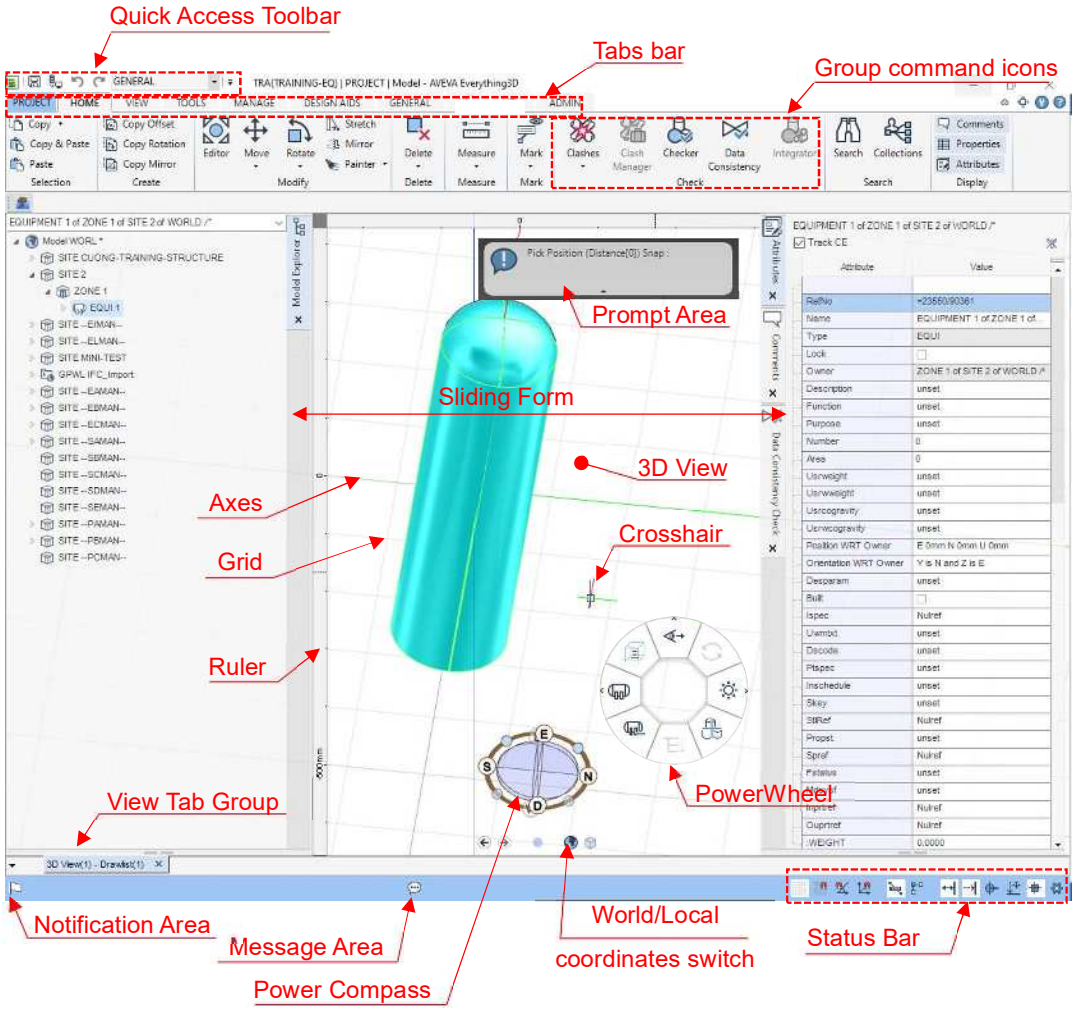
⇨ Select **TRA** from **Project Credentials tiles** > Select correct User > enter passwords > select MDB > click Model Icon 

to enter the AVEVA E3D Model session.

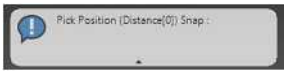
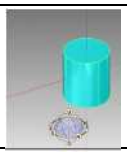
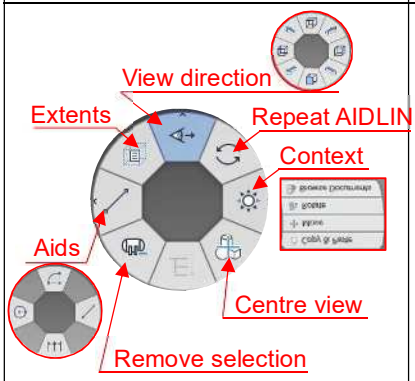
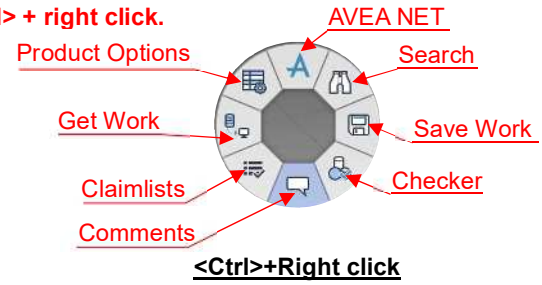
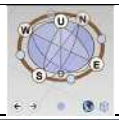
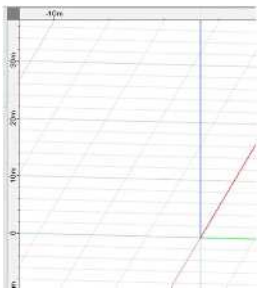
The **AVEVA E3D** initialization animated image will be displayed for a few seconds while the module is loaded along with the databases from the selected MDB.

1.2 The Model Environment

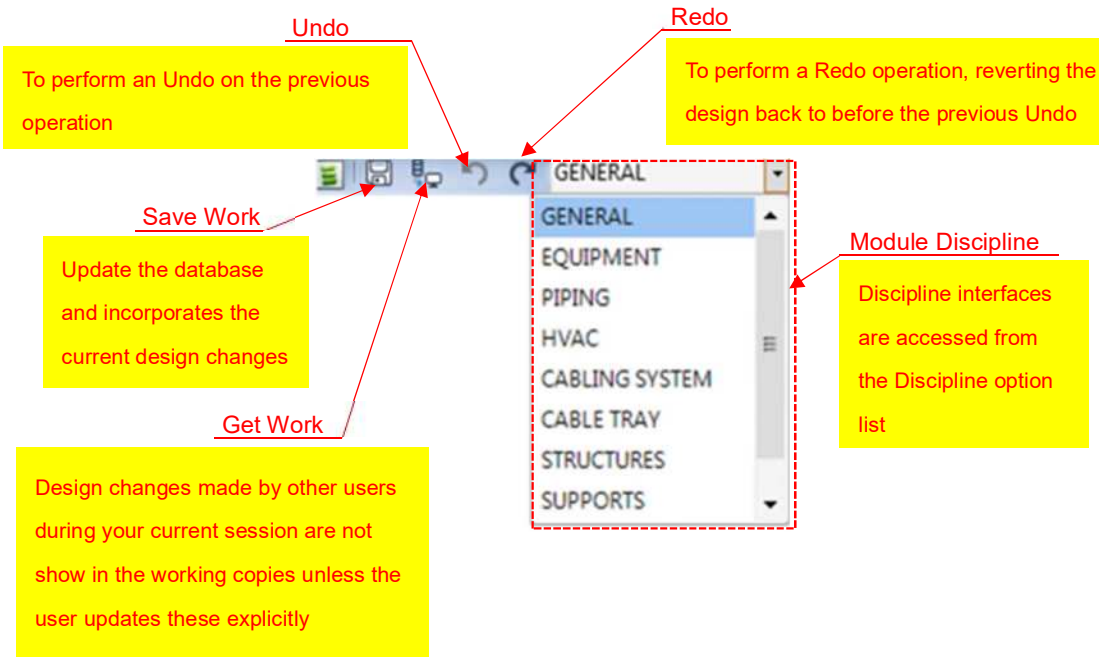
Starting Model for the first time will user the default screen layout as show below:



Explanation:

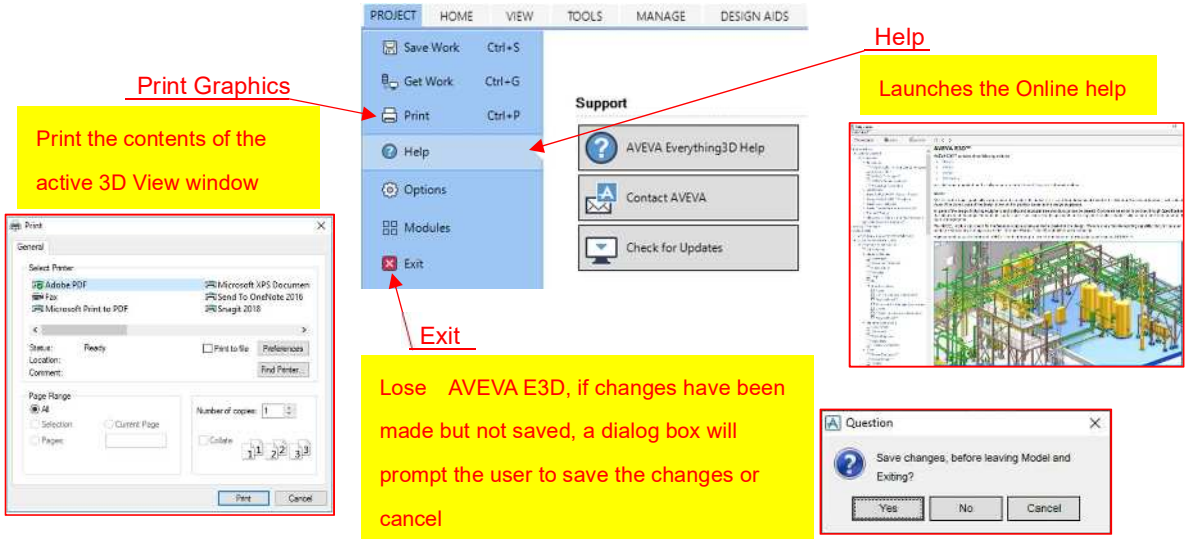
	**Quick Access Toolbar:** provides shortcuts to most common command for a quick access.
	**Tabs Bar**: regroups the application buttons by category. Displayed tabs depend on the selected Discipline.
	**Groups**: each tab is divided into Group to organize the buttons by task
	**Promp Area**: provides instructions while performing an action
	**3D view**: workspace where the Model elements can be viewed and work on.
	**Sliding forms Set or Panels:** from are docked and tabbed on both sides of the 3D View and can be temporarily hidden to clear the 3D view.
	**PowerWheel:** provide quick access to most common commands directly in the 3D View by a **right click**, or extension to the primary PowerWheel by a **<Ctrl> + right click.** 
	**PowerCompass:** provides intuitive tool to orientate the view and select different Coordinate System.
	**Status bar:** provides quick access to common modelling option
	**Message Area:** displays the last information, warning or error message.
	**Notification Area:** displays information about the status of a global project.
	**View Tab Group:** provides access to the different opened 3D View from multiple tabs.
	- **Grip**: displays grip lines on the Working plane. It may be active with the button in the status bar or pressing the <F7> key. - **Ruler:** offers a visual indication of the spacing and the position of the Crosshair projected on the Grip. - **Crosshair:** is the appearance of the cursor when in the 3D, showing a representation of the 3 axes direction. It can be configured in the Project Option. - **Axes:** the LCS axes are represented in the 3D View by 3 colored lines passing through the origin of the LCS.

1.3 Work with Quick Access Toolbar:

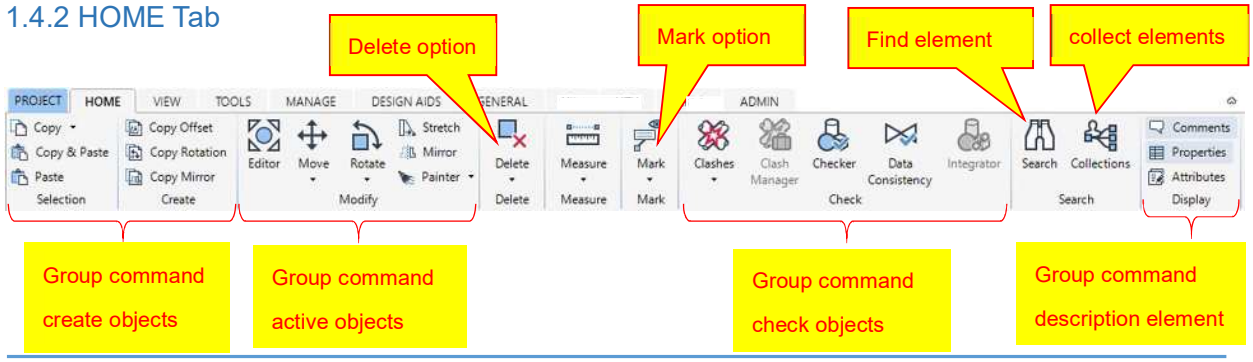


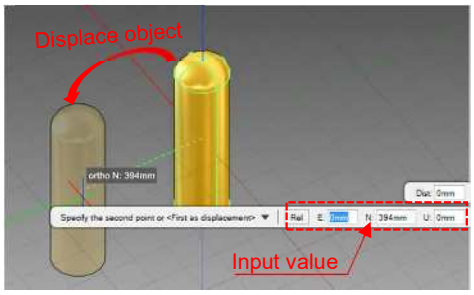
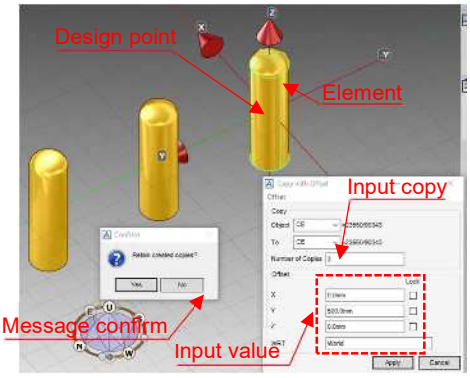
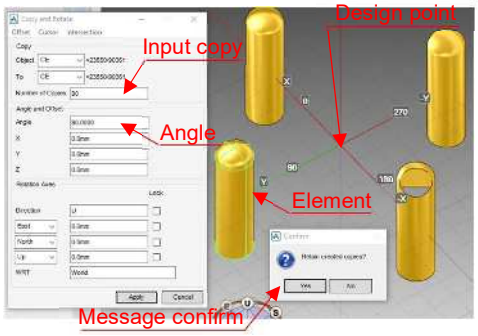
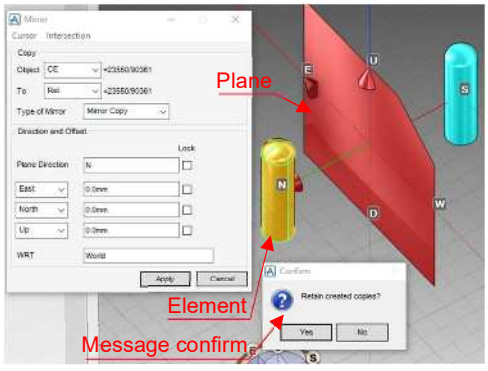
1.4 Work with Tab Bar

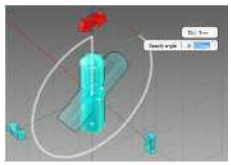
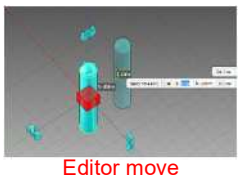
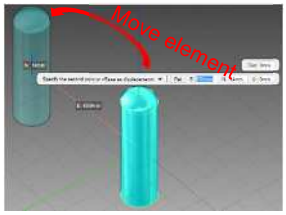
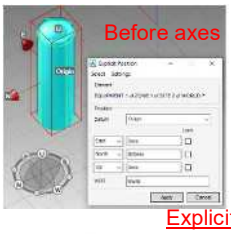
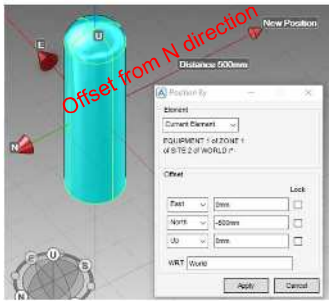
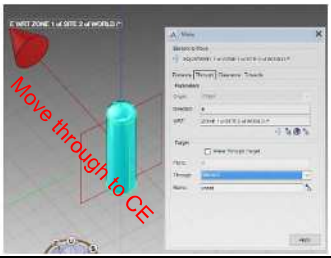
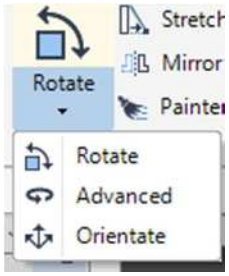
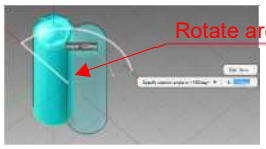
1.4.1 PROJECT Tab



1.4.2 HOME Tab



Function	Description	Image
Copy Copy & Paste Paste Selection	Copies objects to a specified position: - The user are prompted to select object to copy. - Click to identify the objects to copy and press **Enter**. - The user are prompted to **Specify base point or <Displacement>** - Input values in the dimension fields and press Enter or click to identify the new location for copy objects.	
Copy Offset	Creates copies of part of the model and offsets each copy from previous one by a specified distance in a specified direction: - Element: defined as the origin of an element. - Design Point: a p-point or construction pin - Pline: a pline of structural section - Edge: a panel edge linking two vertices.	
Copy Rotation	Create copies of part of model. Each copy can be rotated and offset relative to the previous one.	
Copy Mirror	Create a copy of part of the model and mirrors the copy in a specified plane. The original parts of the model can either be retained or removed.	

	The user to reposition selected item using the mouse like moving, rotating or modifying dimensions.	 Editor rotate  Editor move
	Move the selected object to a new position: ➔ **Position:** element can be explicitly positioned with respect to the world axes or with respect to axes of another element in the hierarchy. ➔ **Offset:** move an element to a specified distance in a specified direction, relative to its current position. ➔ **Advance:** moves the current element (CE) in a given direction. CE can be moved by a given distance in that direction or it can be moved to a position relative to another element or relative to a plane through a given position.	   Explicitly position  
	Rotate an element about an axis ➔ **Advance:** rotate an element through a specified angle about a specified axis. ➔ **Orientate:** The axes (fixed reference lines) of an element can be explicitly orientated or the axes of an existing element can be used.	  

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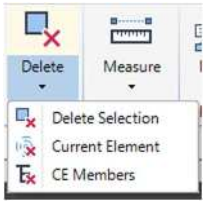
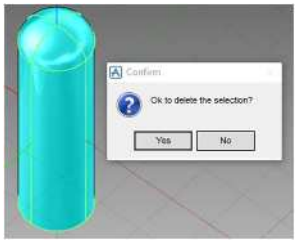
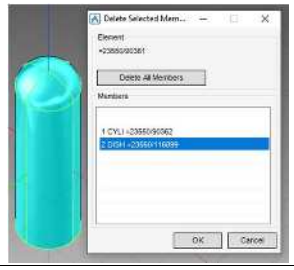
<td> 
</td>
<td>There are several methods for deleting specified elements from the database.
➔ Delete Selection: click this button, or pressing the <Delete> key, will be delete all the elements included in the current graphical selection from the database.
➔ Current Element: click this button will be delete CE from.
➔ CE Member: display a form to select the members of the CE to delete.</td>
<td> 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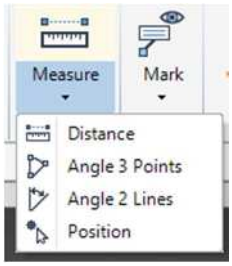
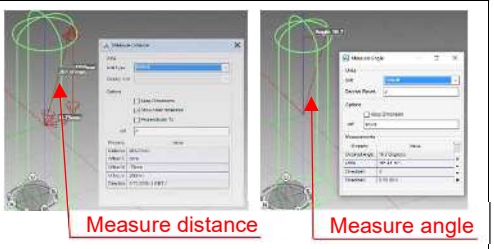
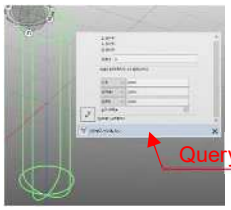
<td> 
</td>
<td>➔ Measure Distance: Measures the distance between two points without having to calculate it.
➔ Measure Angle: measure the angle and the direction of the two angular dimension lines without having to calculate it.
➔ Query Position: queries the position of a point and it’s position wrt to an element which can specified.</td>
<td> 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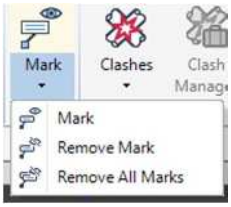
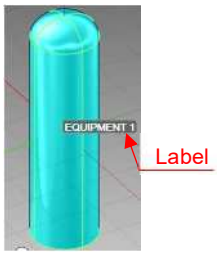
<td> 
</td>
<td>➔ Mark: marks item with a label in 3D graphical view.
➔ Remove Mark: removes the labels from the selected item.
➔ Remove All Mark: remove all the labels from all of the item.</td>
<td> 
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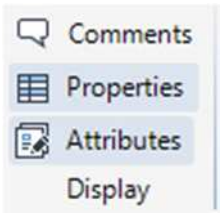
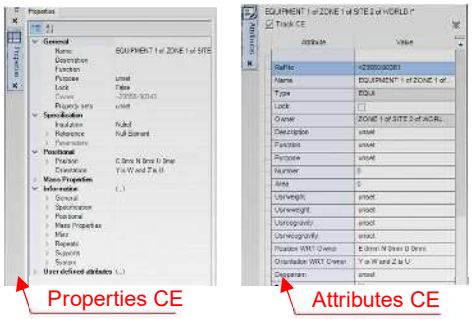
<td> 
</td>
<td>Comments: allow the user to inspect and review the design in real time and provide comments and suggestions for improvements or changes.
Properties: Displays the properties of objects, if the selected objects are read only (locked, claimed or unavailable through direct access control) the properties are greyed out.
Attributes: display the attributes that have been specified for the current selected element (CE), dependent upon the user access rights, some attributes can be modified.</td>
<td> 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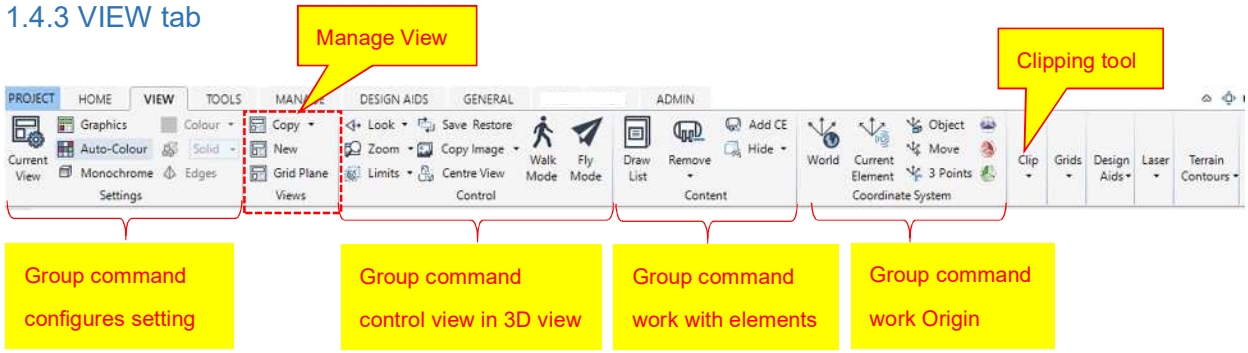
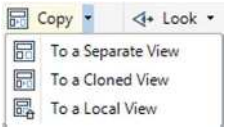
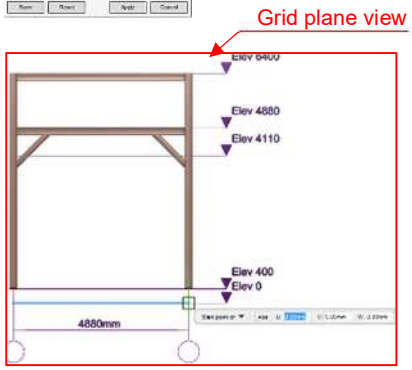
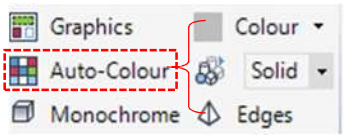
1.4.3 VIEW tab

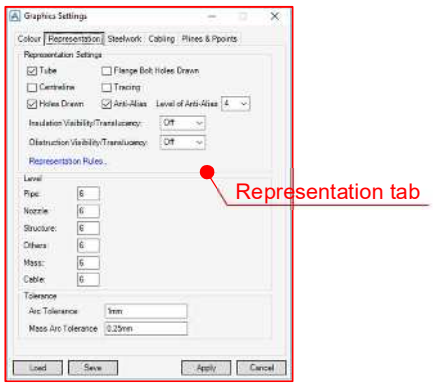


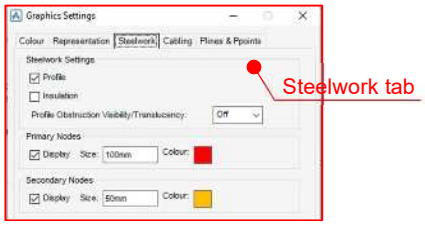
Function	Description	Image
  	➔ **Current View:** configures default settings for current 3D View and uses these settings for any new view created (except Cloned view). Changes made are applied instantly. ➔ **New view:** create an empty view of the 3D Graphical window on a new tab, with the View Settings window displayed and an empty drawlist. ➔ **Grid Plane view:** creates a plane view from the currently selected grid plane of a reference grid. By, default the grid plane view that displays all objects associated with the grid plane. ➔ **To a Separate View**: creates a separate view of the current view on a new tab using the same elements of the Drawlist as the current view. ➔ **To a Cloned View**: creates a clone of the current view, on a new tab, using the same drawlist as the current view, which can be the global drawlist or a local drawlist depending on how the current view itself was created. ➔ **To a Local View**: creates a new local view of the graphical section.	 

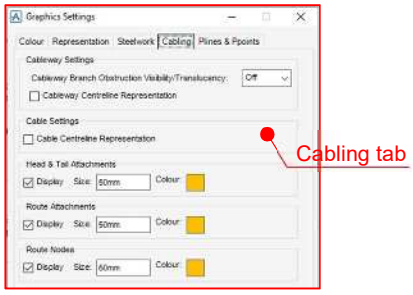


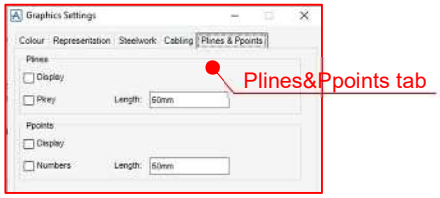
Graphics: Configures the default display option for general 3D graphics, colors and display representation of pipework, steelwork, cabling and Plines and Ppoint.

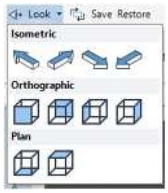
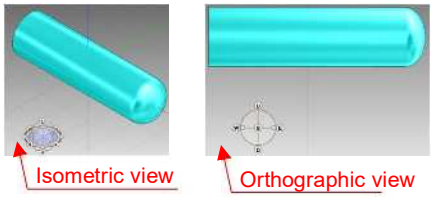
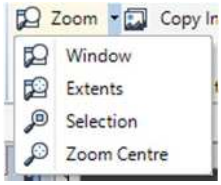
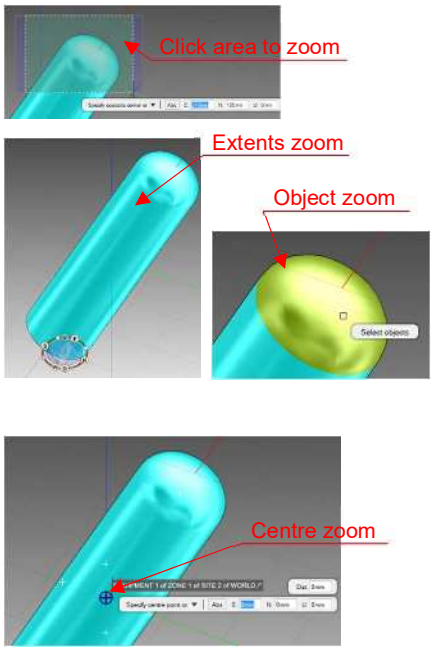
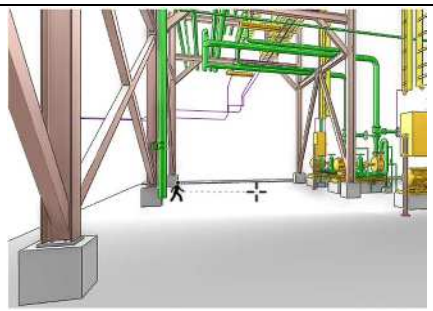
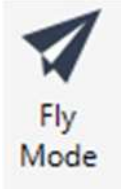
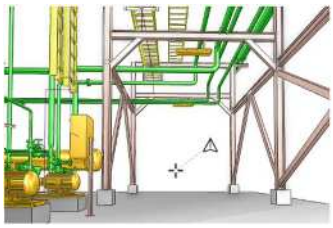
- **Colour tab** sets the defaults colours for the current element, the active element, other visible elements and the colours used for aid lines and highlighting.
- **Auto colour settings**: check box to assign the colours to significant elements when they are added to the Drawlist.
- **Representation tab** of the graphics Settings widow allow the user to set some the general and piping representation parameters and presents control options allowing the user to display further windows for specifying the representations of more specific parameters.
- **Steelwork:** allows the user to change the way Steelwork elements are display.
- **Cabling tab** allows the user to change various display settings for Cabletray elements.
- **Plines and PPoints:** representation for structural Profiles can be set on display or off display. By default Plines are not displayed, although ppoints is displayed automatically as part of an identification operation.

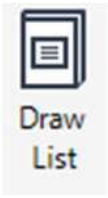
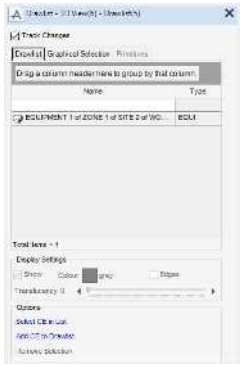
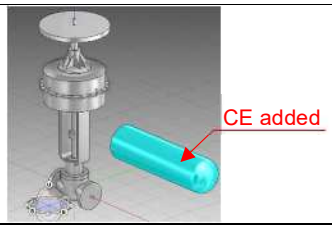
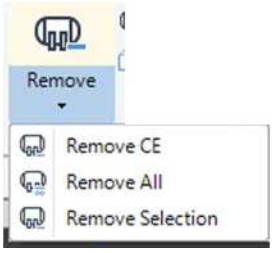
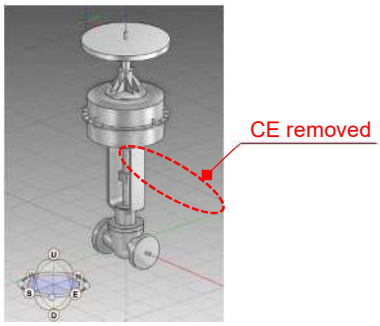
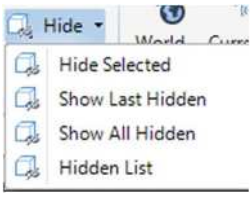
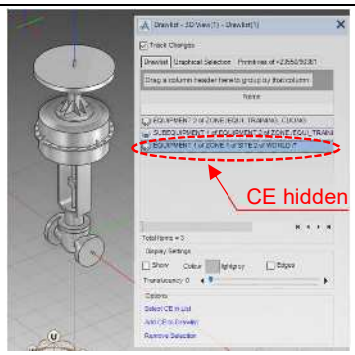


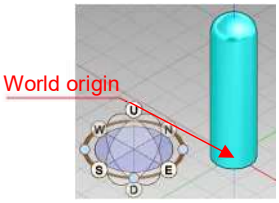
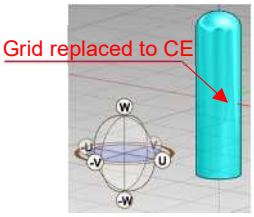
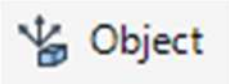
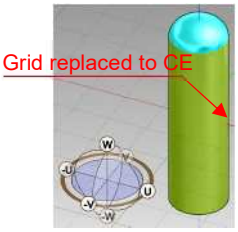
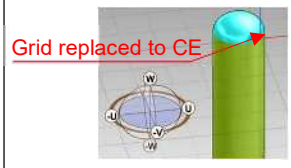
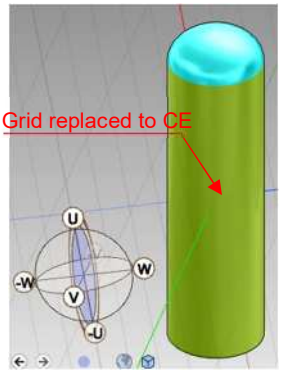
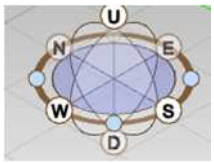
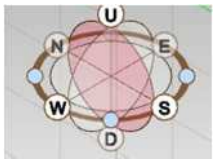
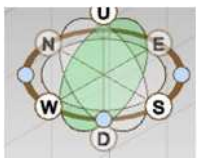


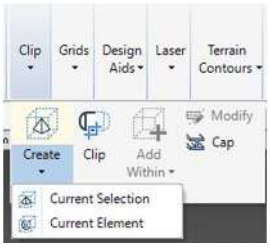


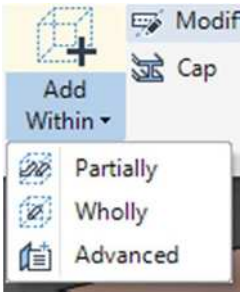


	**Look:** changes the orientation of the view to a preset view, either to an Isometric view, an Orthographic view or Plan view.	
	Zoom: increases or decreases the magnification of the view without changing the size of any objects. ➔ **Window**: zoom to an area of the view defined by a rectangular window (double click in the 3D view) ➔ **Extents**: decreases the magnification of the view to include all of the objects in view. ➔ **Object**: zoom to display the selected object in the middle ò the window. ➔ **Centre**: zoom to an area of the view defined by picking a center point on the view and magnification value.	
	**Centre View:** pans the view to position the current element in the center of the view window.	
	**Walk Mode:** navigates within the model in any direction on a horizontal plane. ➔ Press Shift: shilst moving to increase speed of movement. ➔ Press Ctrl shilst moving to decrease the speed of movement. ➔ Depress the middle mouse button to rotate around a fix position.	
	**Fly Mode:** navigates freely through the model in any direction.	

	**The Drawlist Form:** allows the visualization of the list of elements contained in 3D View. From this form, all the Add and Remove options seen previously are available, along with additional options to modify the appearance of the model elements in the 3D View, like colours and translucency.	
	**Add CE:** adds only the current element (CE) to the drawlist.	
	➔ **Remove CE:** removes only the current element from the Drawlist. ➔ **Remove All:** remove all the elements from the Drawlist. ➔ **Remove Selection:** remove all selected elements from the Drawlist.	
	➔ **Hide Selected**: hides the selected elements in the 3D view but does not remove it from the Drawlist. ➔ **Show last Hidden:** displays the element most recently hidden during the current session, provide the element is still in the Drawlist. ➔ **Show All Hidden:** display all the hidden elements during the current session. ➔ **Hidden List**: lists all the elements in the Drawlist for the current view that are currently hidden.	

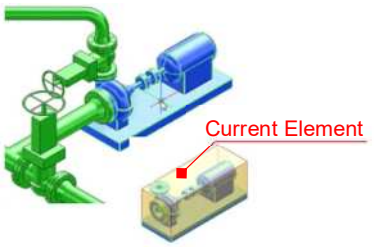
	**World:** sets the origin at (0,0,0) world position, and an orientation set to East, North and Up axes.	
	**Current Element:** set the axis at the Point of Origin of the CE with the grid parallel to CE orientation.	
	**Object:** Sets the axis at the Point of Origin of the selected object with the grid parallel to the selected object orientation. If no object is selected you will be prompted to select one	
	**Move:** Moves the Point of Origin but does not changes the orientation.	
	**3 Points:** Prompts you to pick 3 points which are used to define the position of the new XY plane. The three points represent three positions in the new XY plane. The first point will become the origin of the new local coordinate system. The second point can be any point on the positive portion of the X axis. The third point can be any point on the positive portion of the Y axis	
	**EN:** Sets the axis to display from U to V	
	**NU:** Sets the axis to display from V to W	
	**EU:** Sets the axis to display from U to W	

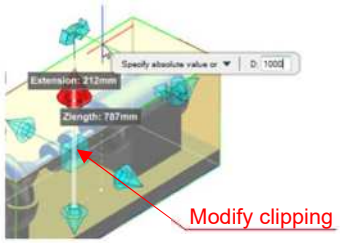


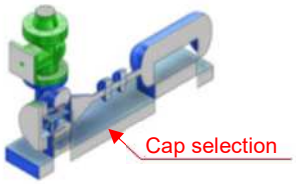


Clipping the View: allows the user to isolate a region of the model by hiding all parts of the model falling outside a user defined clipping box.

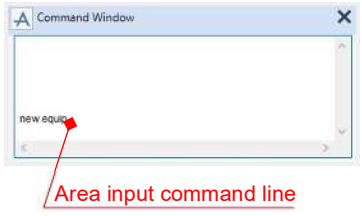
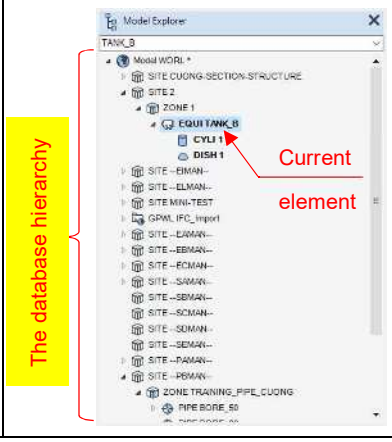
- ➔ **Current Selection:** select from the crop-down list to create a clipping box with limits defined from a selection of elements.
- ➔ **Current Element:** Select from the drop-down list to create a clipping box with limits defined from the currently selected element
- ➔ **Clip:** to turn clipping on or off.
- ➔ **Add Within:** to define the elements which are contained within the clipping box.
  - ⇨ ***Partially***: Select from the drop-down list to display all elements in the 3D view which are partially contained within the clipping box
  - ⇨ ***Wholly***: Select from the drop-down list to display all elements in the 3D view which are wholly contained within the clipping box
  - ⇨ ***Advanced***: Select from the drop-down list to define the elements which are contained within the clipping box
- ➔ **Modify:** Click to turn the clipping box display on or off. Double click with the left mouse button to modify the dimensions of the clipping box as a non-routed item using the **Editor** functionality.
- ➔ **Cap:** When the model is clipped, only the parts of the model inside the clipping box are displayed. Where items are intersected by the clip box, a coloured cap can be added to show that the items extend beyond the displayed region.



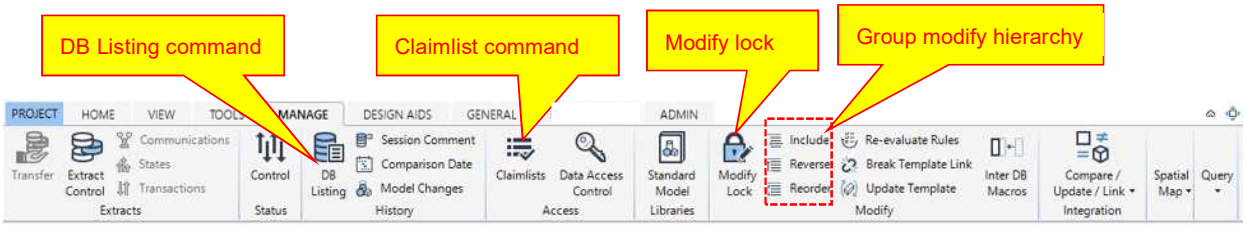


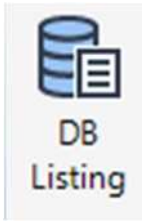
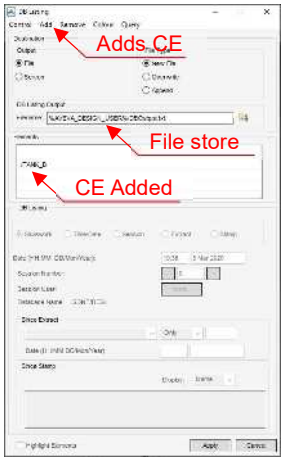
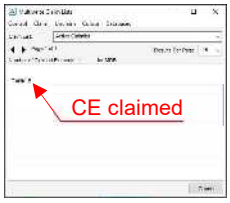
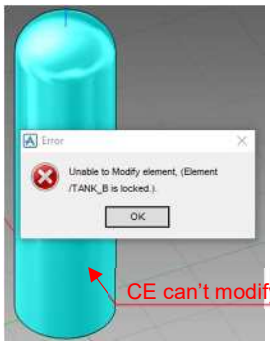
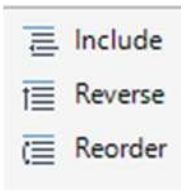
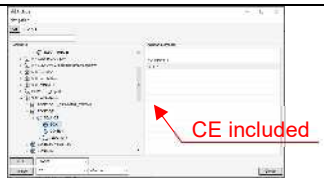
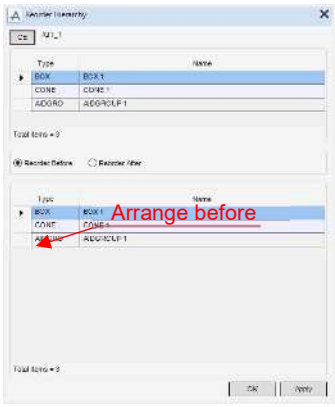


1.4.4 Tool tab

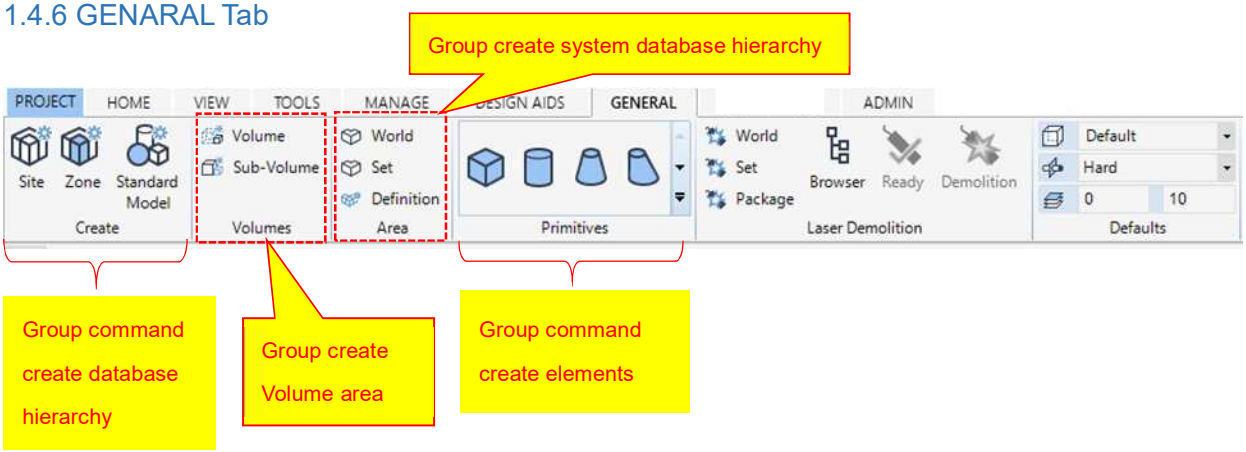
Commands	**Command Window:** the user are able to streamline its methods of working and save time on repeated task. ⇨ Create macros to automate repetitive procedures ⇨ Design and create new windows and menus for the graphical user interface that match the working needs precisely.	
Messages	**Messages:** displays User and System messages. ⇨ User: contain error, warning and information messages. ⇨ System messages: are messages or trace created by the application of no particular type or output from the SYSCOM command.	
Model	**Model Explorer:** provides an easy to use graphical of exploring the Design Database. ⇨ The Model Explorer displays the database hierarchy. Click on an element in the tree view to navigate to the element in the database and make it the current element ⇨ When you navigate to an element by using the command line, a graphical pick, or by any other means, the appropriate part of the tree view is expanded to display the current element.	

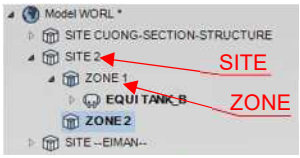
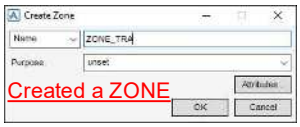
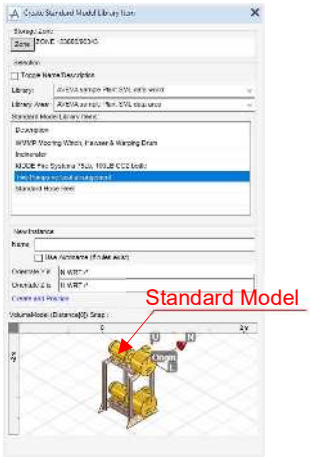
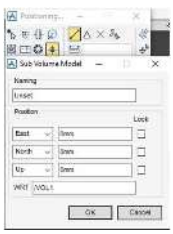
1.4.5 MANAGE Tab

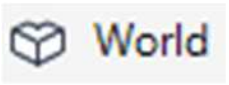
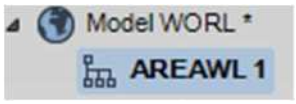
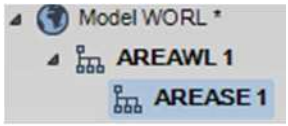
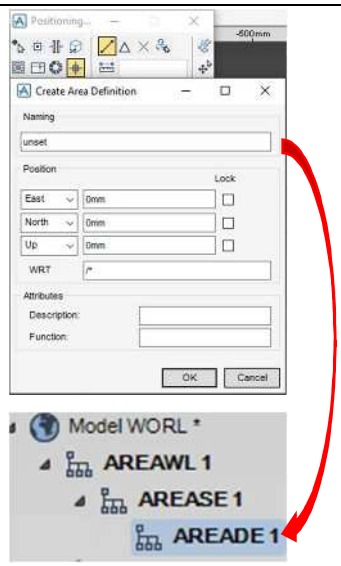
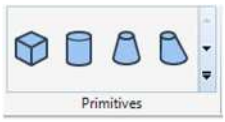
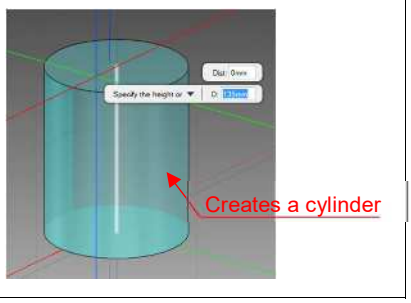


	**DB Listing:** the DB Listing window allows the user to query and output aspects of the database for example to report changes since a certain date.	
	**Claimlist:** use to claim elements in a multiwrite database. Claiming an element in a database stops other users working on it. These user can't write to it.	
 	**Modify lock:** Locks or unlocks any design element. Locking an element prevents it from being modified or deleted. No changes to the element are then possible until it is subsequently unlocked. ⇨ **Lock CE:** locks current element only ⇨ **Lock CE and below**: locks current element and its members. ⇨ **Unlock CE**: unlocks current element only ⇨ **Unlock CE and below**: unlocks current element and itsmember.	
	➔ Include: moves elements from the members list of one owner to that of another. ➔ Reverse Order: reverses the entire members list order of an element. ➔ Reorder: rearranges the members list order for any owning element.	 

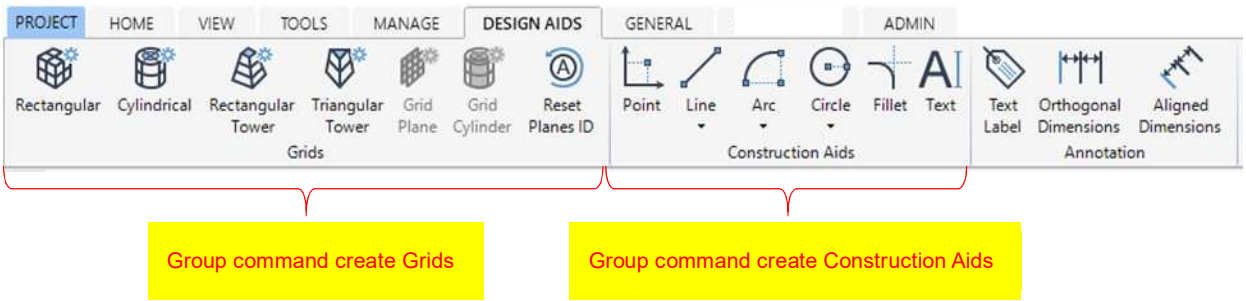
1.4.6 GENARAL Tab



Site Zone	➔ Creates a Site. Before any element can be created, a Site and a Zone must be created. These two elements are a way of grouping item in specific areas to allow greater control of the model. ➔ A SITE is owned by the WORLD and the ZONE is owned by the SITE. The SITE, is the entire area and holds a number of ZONE elements, these can be all the zone within the site.	  
Standard Model	**Standard Model Library Manager:** Captures multi-discription elements and stores them in a library for re-use as Standard Model Library Item.	
Volume Sub-Volume	➔ **Volume:** creates a Volume (VOLM) which can be used to represent items or to dedicate space for future use. ➔ **Sub Volume**: creates a Sub-Volume (SVOLM) which can be used to divide Volume Model into sub-parts.	 Creates Volume  Creates Sub-Volume

	**Create Area World:** creates an area world database element. An area world element is an administrative element that can own a number of area set elements.	
	**Create Area Set:** creates an area set database element. An area set element is an administrative element that can own a number of area set or area definition elements.	
	**Create Area Definition**: creates an area definition database element. An area definition element is a container element that can own primitives that define area volumes.	
	**Create Primitives:** primitive item are the design date elements (ex: box, cylinder,..) that allow the user to build up a graphical representation of the equipment item.	

1.4.7 DESIGN AIDS Tab



Rectangular

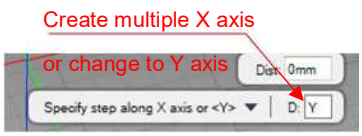
Create Rectangular Reference Grid: which contains two horizontal faces (on the X and Y axis) and an option additional vertical face (on the Z axis).

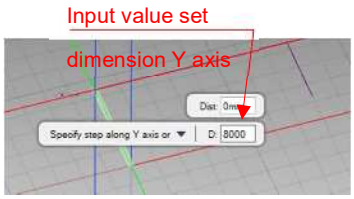
- ⇨ The user prompted to **Specify grid origin** or input values in the dimension fields and press **<Enter>**
- ⇨ The user are prompted to **Specify Step along X** axis or input a dimension value in the field and press **<Enter>**
- ⇨ The user prompted is repeated (allowing the user to create multiple planes along the X axis) or input **<Y>** at the prompt and press**<Enter>** to create the Y axis of grid.
- ⇨ The user are prompted to **Specify Step along Y** axis or input a dimension value in the field and press **<Enter>**
- ⇨ The user prompted is repeated (allowing the user to create multiple planes along the Y axis) or input **<Z>** at the prompt and press**<Enter>** to create the Z axis of grid.
- ⇨ The user prompted to Specify Step along Z axis or input a dimension value in the field. The prompt is repeated or press **<F> + <Enter>** or **<Esc>** to the end the command.

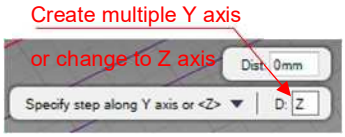
Shortcut key		
Key	**Meaning**	**Description**
N	Name	To define the grid name
A	Absolute	To define a grid plane with a distance value measured from the grid origin
B	Back	To undo the last action
R	Relative	To define a grid plane with a distance value measured from the previously created grid plane
Y	Y	To create the Y axis of the grid
Z	Z	To create the Z axis of the grid
F	Finish	To end the command

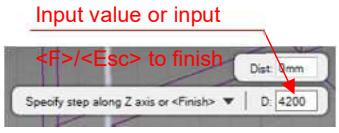
To modify grid: To input <IN> at command line and prompted to select element.

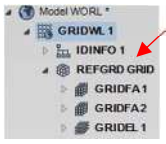
- ⇨ Changed the information in property board.

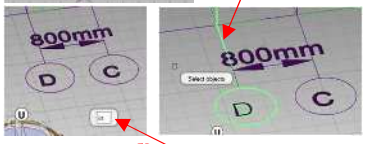


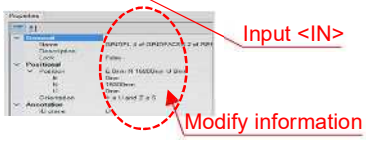


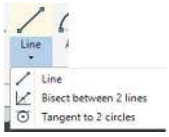
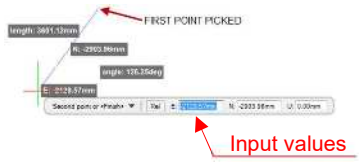
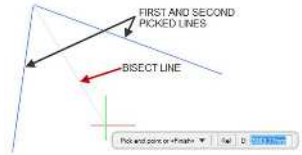
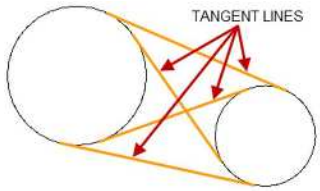
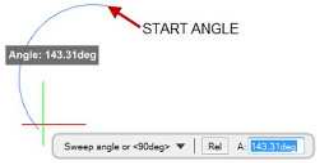


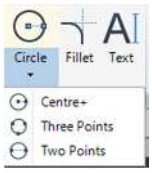
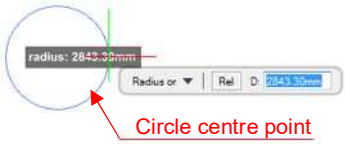
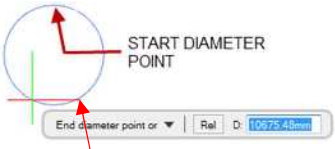








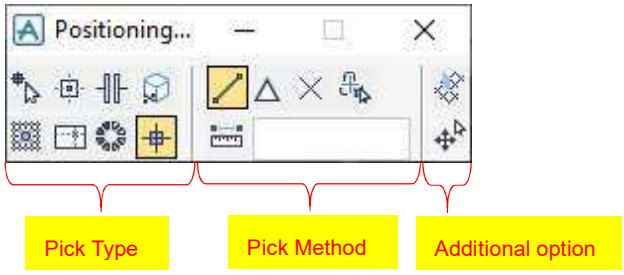
	**Point:** creates a construction aid point which can be positioned using a cursor pick on the 3D view. The user are prompted to pick the Origin point. Input values in the dimension field and press **<Enter>** or click to identify the construction point position.	
	➔ **Line:** creates a construction aid line between two specified points. ⇨ The user are prompted to pick the **First point** or **<finish>**. Input value in the dimension fields and press **<Enter>** or click to identify the first point position. ⇨ The user are prompted to pick the **Second point** or **<finish>**. Input value in the dimension fields and press **<Enter>** or click to identify the second point position. ➔ **Bisect between Two Lines:** creates a construction aid line bisecting two specified construction aid lines. The start of the line is positioned at the intersection of the two picked lines. ⇨ The user are prompted to **Pick first line**. The user click to identify the first construction aid line using a cursor pick in the 3D view. ⇨ The user are prompted to **Pick Second line.** Click to identify the Second construction aid line using a cursor pick in the 3D view. ⇨ The user **Pick end point** or **<Finish>** ➔ **Tangent to Two Circle:** creates a construction aid line tangent to two specified construction aid circles. A tangent line can be created in four position, dependent on the area of each circle that is selected during the creation process.	  
	➔ **Arc Centre + Sweep:** created an arc specified centre point, radius and sweep angle. ⇨ The user are prompted to pick **Centre point.** Input values in the dimension fields and press **<Enter>** or click to identify the centre point position. ⇨ Then prompted to **Radius.** Input a dimensional value in the field and press **<Enter>** or click to identify the radius point position.	

	⇨ Then prompted to **Start Angle or <#deg>.** Input an angular value in the field and press **<Enter>** or click to identify the start point position. ⇨ Then prompted to **Sweep Angle or <#deg>.** Input an angular value in the field and press **<Enter>** or circle to identify the end point. ➔ **Arc three Point:** creates an arc with three specified radius points. ⇨ The user are prompted to pick the **1st circumference point.** Input values in the dimension field and press **<Enter>** or click to identify the first point position. ⇨ Then prompted to pick the **2nd circumference point.** Input values in the dimension filed and press **<Enter>** or click to identify the second point position. ⇨ Then are prompted to pick the **3rd circumference point**. Input values in the dimension filed and press **<Enter>** or click to identify the third point position.	
	➔ **Circle Centre Point:** creates a circle with a specified centre point and radius/diameter point. ⇨ The user are prompted to pick the Centre point. Input values in the dimension field and press <Enter> or click to identify the centre point position. ⇨ Then prompted to pick the **Radius or**. Input a dimensional value in the field and press **<Enter>** or click to identify the radius point position. ➔ **Circle Three Points:** creates a circle with three specified circumference points. ⇨ The user are prompted to pick the **1st circumference point.** Input values in the dimension fields and press **<Enter>** or click to identify the first point position. ⇨ The user are prompted to pick the **2nd circumference point**. Input values in the dimension fields and press **<Enter>** or click to identify the second point position. ⇨ The user are prompted to pick the 3rd **circumference point**. Input values in the dimension fields and press **<Enter>** or click to identify the third point position.	 Circle centre point  Circle three points  Circle two points

	➔ **Circle Two Points**: creates a circle with two specified points. ⇨ The user are prompted to pick the **Start diameter point.** Input values in the dimension field and press **<Enter>** or click to identify the start point position. ⇨ Then prompted to pick the **End diameter point.** Input a dimensional value in the field and press **<Enter>** or click to identify the end point position.	
	**Fillet:** creates a fillet from two specified construction aid lines. ⇨ The user are prompted to **Pick first line.** Click to identify the first construction aid line using a cursor pick in 3D view. ⇨ Then prompted to **Pick second line**. Click to identify the second construction aid line using a cursor pick in 3D view ⇨ Then prompted to enter the required **Radius**. Input a dimensional value in the field and press <Enter> to define the radius of the construction aid fillet circle.	
	**Text**: creates text. ⇨ The user are prompted to pick the position. Input values in the dimension fields and press **<Enter>** or click to identify the text start position. ⇨ Then prompted to select the Text height. Input a dimensional value in the field and press **<Enter>** or click to identify the text height. ⇨ Then prompted to select the **Text orientation or <#deg>**. Input an angular value in the field and press **<Enter>** or click to identify the text angle. ⇨ Then prompted to enter the Text or. Input the construction aid text to displays in the 3D view in the field and press **<Enter>**	

The positioning Control:

Some commands will be required a different method to specify positions called the Positioning Control, also referred as to Event Driven Graphics (EDG)



Pick Type		Pick Method		Additional option	
	Element: picking is restricted to elements origin		**Snap**: selects the snap point nearest to the cursor pick point.		**Work Plane**: this button activates the working Plane if one has been derived.
	Ppoint: picking is restricted to Ppoints or panel vertices.		**Mid-Point:** derives the mid-point between two snap points along a linear item		**Explicit Position:** the button displays the Explicit Position from allowing explicit co-ordinates to be entered.
	Pline: picking is restriced to structural Plines.		**Intersect:** derives the intersection of two picked lines from any directional element.		
	Graphics: enables adge, surface and corner picks on any graphical element.		**Cursor:** places the derived point exactly where the cursor picks on the element.		
	Aid: picking is restricted to 3D Aid constructs, including User Grid Systems.		**Distance**: applies the offset value entered in the textbox.		
	Screen: enables a pick anywhere in 3D View which identifies two co-ordinates, the third co-ordinate being taken from the current Working Plane or normal to the view direction if there is no active Work Plane.				
	Laser: picking is retricted to Laser data points.				
	Any: uses any appropriate pick type.				

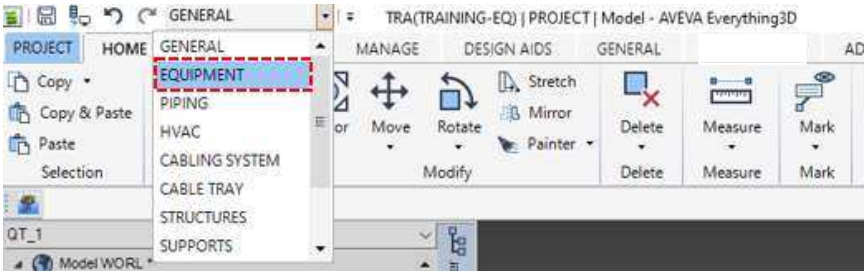
Common shortcut key:

Function key	Description
F1 or 	Online help
F2 or 	Clear Canvas
F3 or 	Object Snap On/Off
F4	Object Snap Projection On/Off
F5 or 	Toggle Walk Mode On/Off
F6 or 	Toggle Fly Mode On/Off
F7 or 	Grid On/Off
F8 or 	Orthogonal drawing On/Off
F9 or 	Grid Snap On/Off
F10 or 	Polar Tracking On/Off
F11	Toggle Shaded/Wireline mode
F12 or 	Dynamic hints On/Off
Delete	Deletes selected elements in 3D View
Esc	Exits the current CIE option/ Cancels position input/ Remove selection/ Exits edit mode.
Enter	Confirm position, Selects default option, Executes command.
Ctrl + A	Select all elements in View
Ctrl + C	Copies via Mid Position of all selected elements
Ctrl + F	Displays the Search window.
Ctrl + V	Pastes the contents of the clipboard
Ctrl + X	Cut the selection
Ctrl + Y	Performs a Redo
Ctrl + Z	Perform an Undo
Alt + ← or ↑ or → or ↓	Pans 3D view toward Left/Up/Right/down
Alt + 1 or 3	Zoom Out of 3D View
Alt + 7 or 9	Zoom in of 3D View
Alt + 2/8/4/6	Pans 3D View Down/Up/Left/Right
Left Mouse Click	Select any elements
Right Mouse Click	Invokes the PowerWheel in CIE Mode
Middle Mouse Click	Centre View to selected point
Mouse Wheel Scroll Up/Down	Zoom In/Out

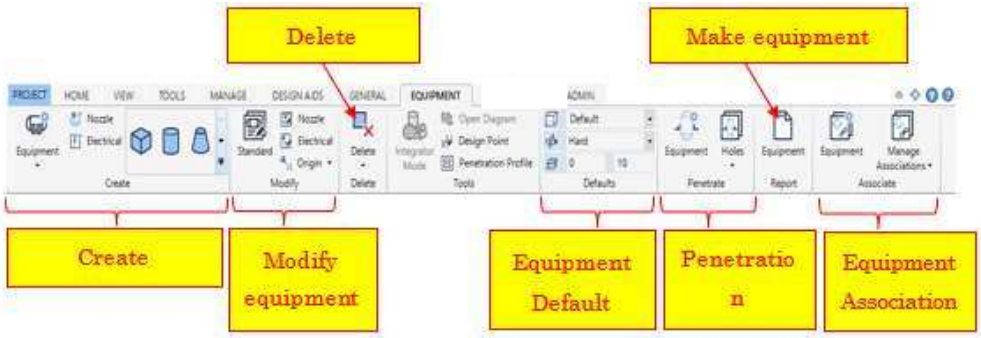
2. Equipment

2.1 Equipment interface

Access equipment module from Quick Access Toolbar:



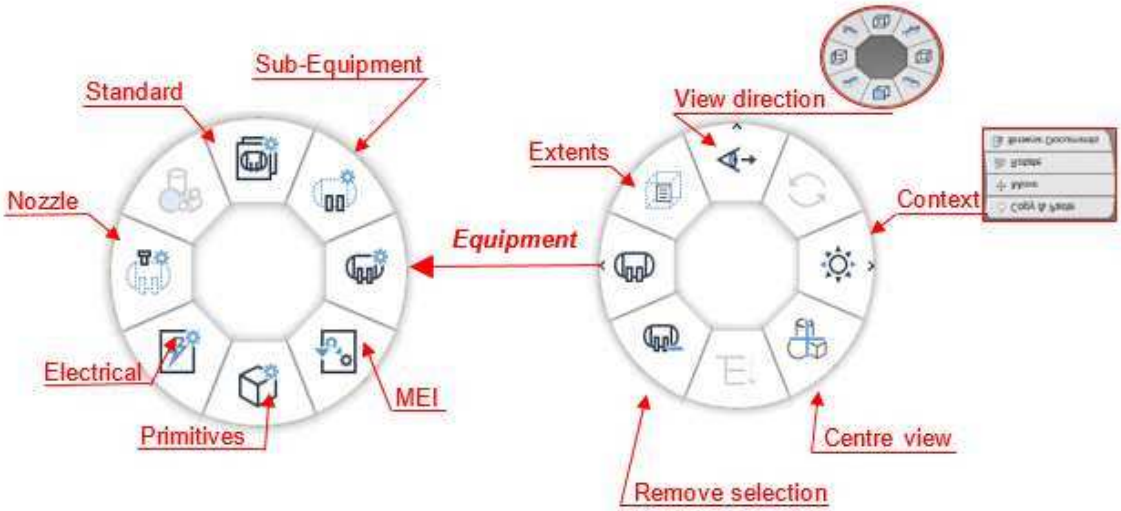
Equipment Toolbar:



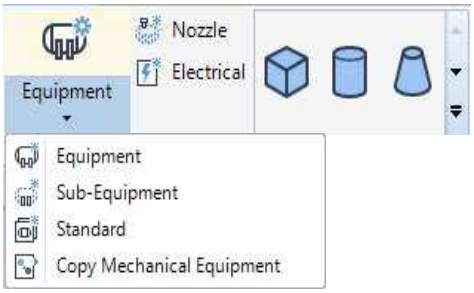
2.2 Work with Equipment

2.2.1 Navigator PowerWheel

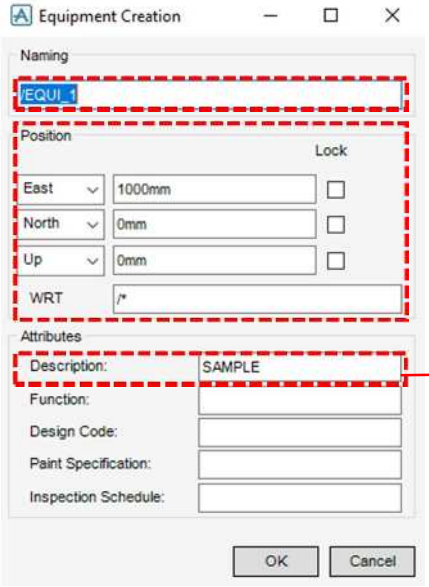
Right-click in 3D view to display the General PowerWheel → click **Equipment** symbol to display Equipment PowerWheel



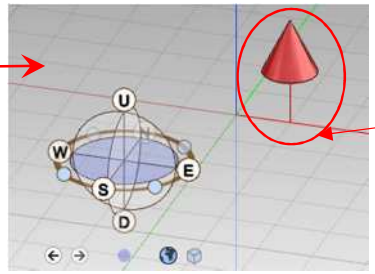
2.2.2 Create group



2.2.2.1 Create equipment

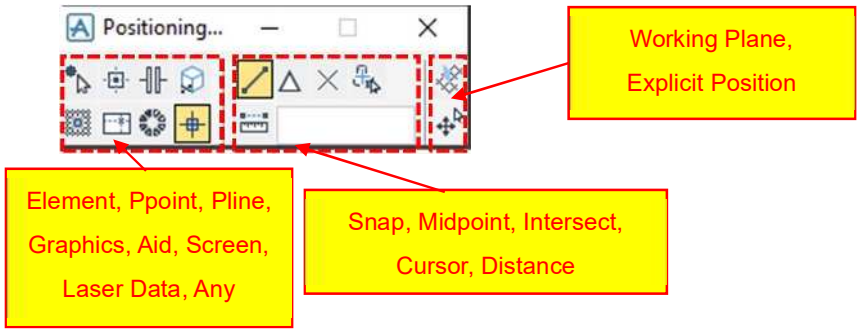


- Enter name of EQUI in the **Naming** field
- Enter **Position** of equipment → the **Position pane** sets the coordinates of the equipment element's origin in world co-ordinates by default or changing to other design elements
- Input attributes of the equipment element in the **Attributes** (Description only necessary) pane

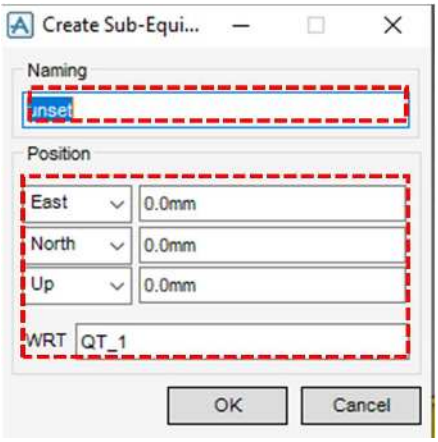


The equipment element can be positioned on one of three ways

- Using the functionality available from the **Position pane** of the **Equipment Creation** window
- Using the cursor and picking a position in the graphical view
- Using the functional available from **Positioning Control** window, refer to the image below for further information



2.2.2.2 Create Sub-Equipment

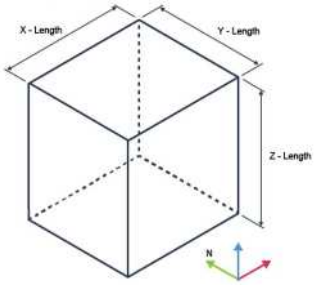


- Enter name of EQUI in the **Naming** field
- The **Position pane** sets the coordinates of the SUBE element's origin and **SUBE** origin is defined **with respected to** the equipment origin (default)
- The **WRT** textbox displays the owning equipment element by default, the user can input an alternative value in the field

2.2.2.3 Primitives

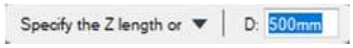
Note:

- Create Equipment and Sub-Equipment (if necessary) first → Create Primitive
- Use and pay attention to Coordinate System as **World**  or **Current Element**  for whole of primitive creating process.
- Use and pay attention to the **Working Plane** as creating primitive profile.
- The lock symbol will be displayed next to each blank after finishing inserting for each one.
  - ➢ *Box Primitive*



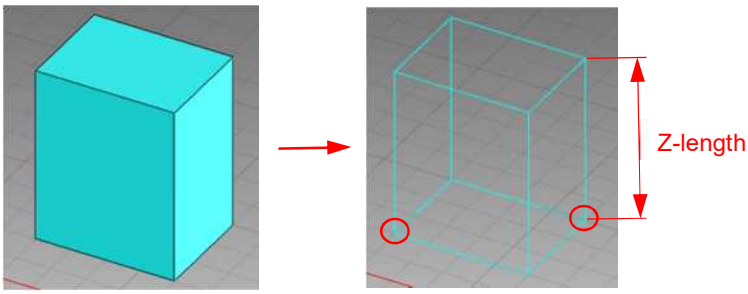
Click **Create Box Primitive** → 

enter values in each axis for the first coordinate → 

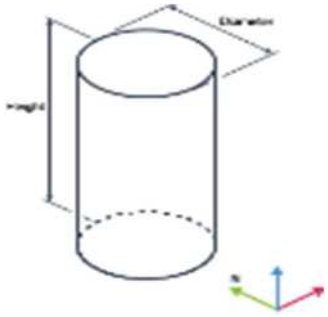
enter values in each axis for the opposite coordinate → 

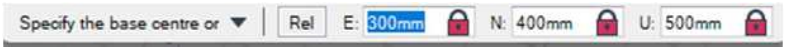
→ **Enter**

The **Box Primitive** created shows as below:



- Cylinder Primitive

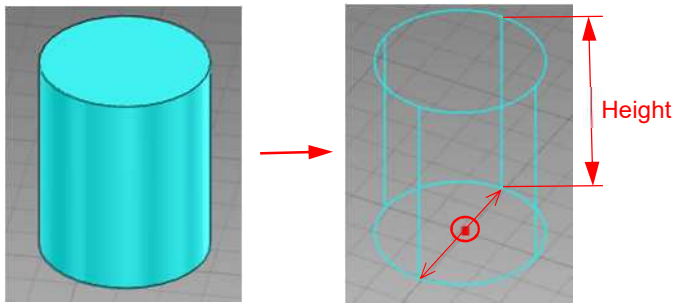


Click **Create Cylinder Primitive** → 

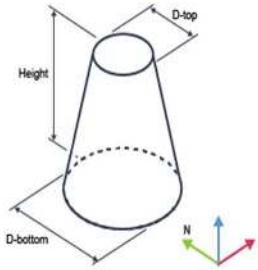
enter values in each axis for the base centre → 

enter value for the diameter → 

input value for the height → **Enter**



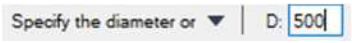
- Cone Primitive



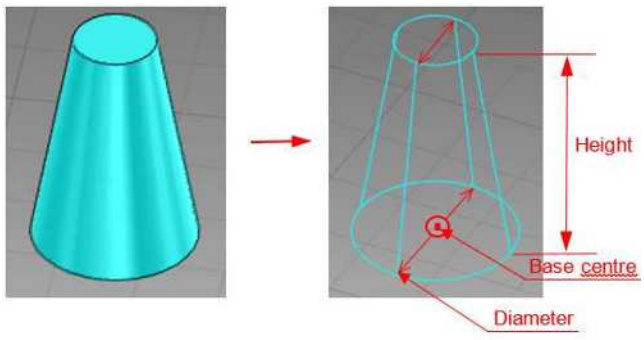
Click **Create Cone Primitive** → 

enter values in each axis for the base centre → 

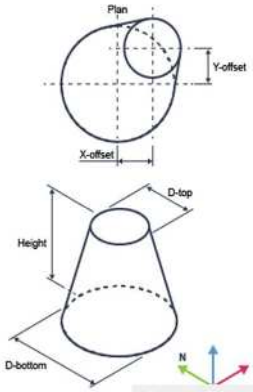
enter value for the diameter → 

Enter value for the height → 

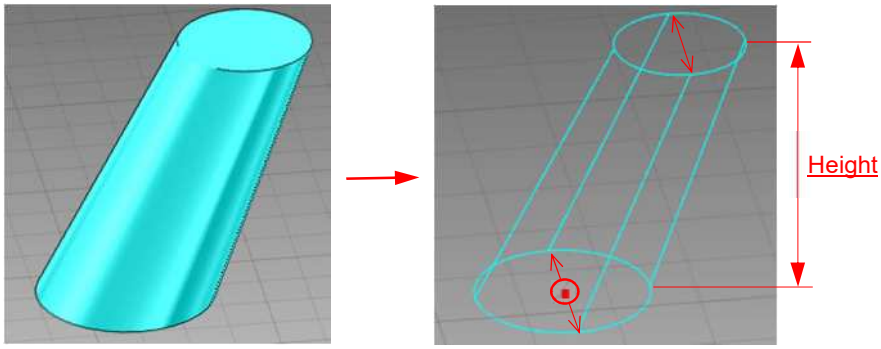
enter value for the diameter → **Enter**



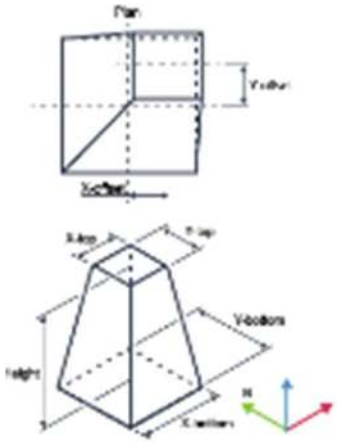
- Snout Primitive



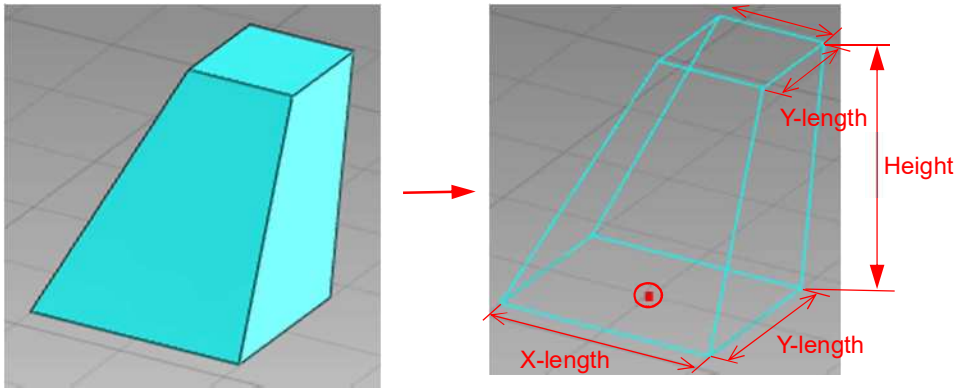
Click **Create Snout Primitive** → Specify base centre or | Rel | E: 500mm | N: 500mm | U: 0mm enter values in each axis for the base centre → Specify the diameter or | D: 2000 enter value for the diameter of **Top Plane**→ Specify the height or | D: 2500 enter value for the height - Specify the diameter or | D: 1500 enter value for the diameter of **Bottom Plane** → Specify the offset or | Rel | E: 1500mm | N: 1000mm enter values for identifying the offset → **Enter**



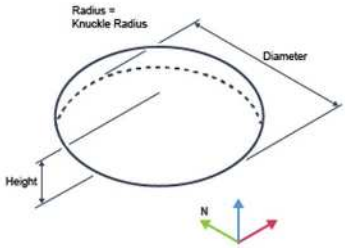
- Pyramid Primitive



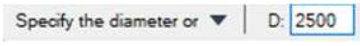
Click **Create Pyramid Primitive** → Specify the base centre or | Rel | E: 1000mm | N: 1000mm | U: 0mm enter values for the base centre → Specify the X length or | D: 2000 enter value for the X length of **Top Plane** → Specify the Y length or | D: 1500 enter value for the Y length of **Top Plane** → Specify the height or | D: 2000mm enter value for the height → Specify the X length or | D: 1000mm enter value for the X length of **Bottom Plane** → Specify the Y length or | D: 1000 enter value for the Y length of **Bottom Plane** → Specify the offset or | Rel | U: 400mm | V: 800mm enter the value for identifying the offset → **Enter**



➢ Dish Primitive

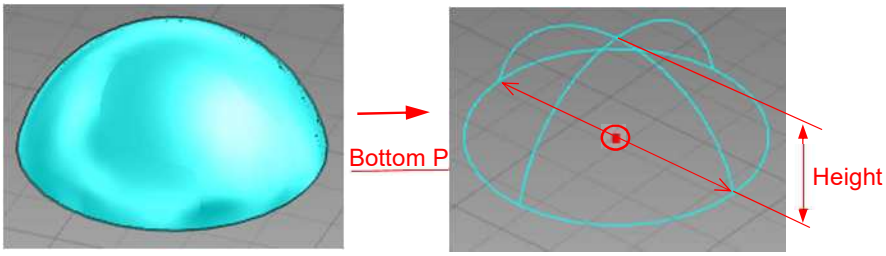


Click **Create Dish Primitive** → 

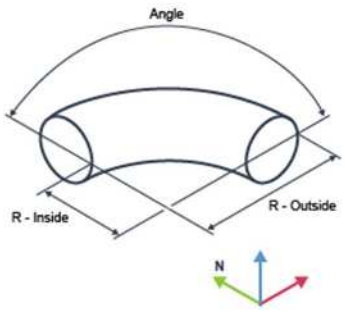
enter values for the base centre → 

enter value for the diameter → 

enter value for height → **Enter**



➢ Circular Torus Primitive



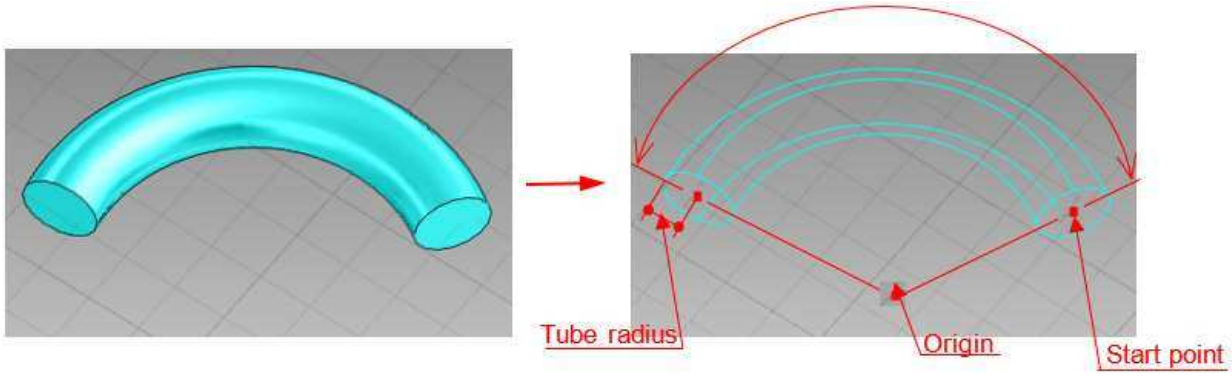
Click **Create Circular Torus Primitive** → 

enter the values in each axis for the origin → 

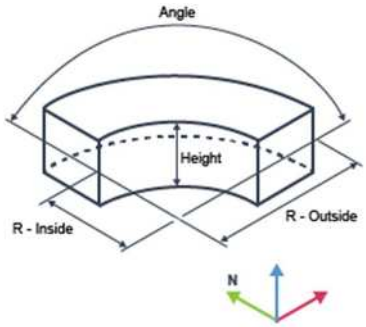
enter values in each axis for the start point → 

enter value for the sweep angle → 

enter value for the tube radius

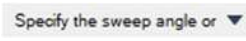


- Rectangular Torus Primitive



Click **Rectangular Torus Primitive** → 

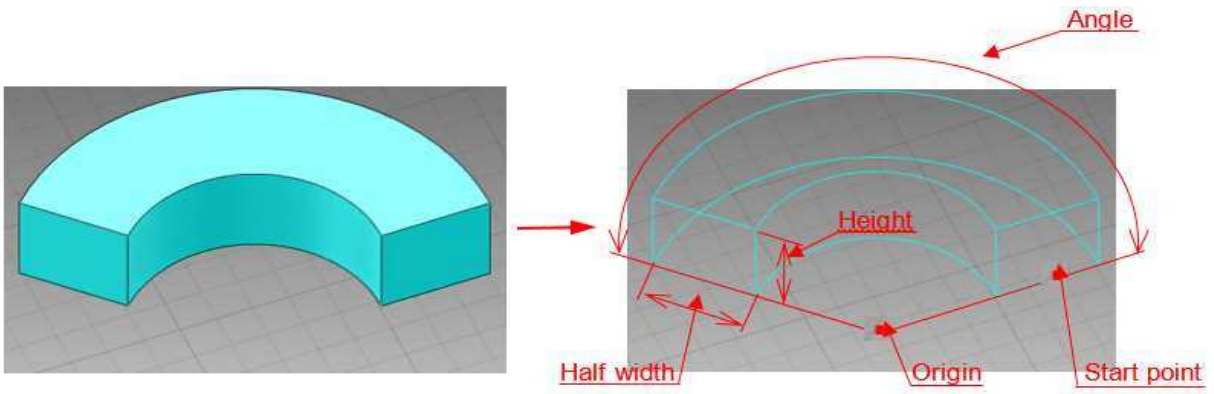
enter values in each axis for the origin → 

enter values in each axis for the start point → 

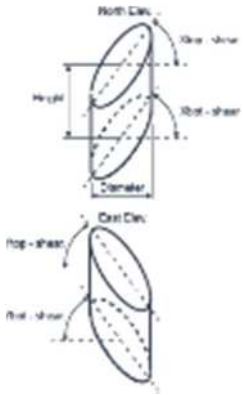
enter value for the sweep angle → 

enter value for the half width → 

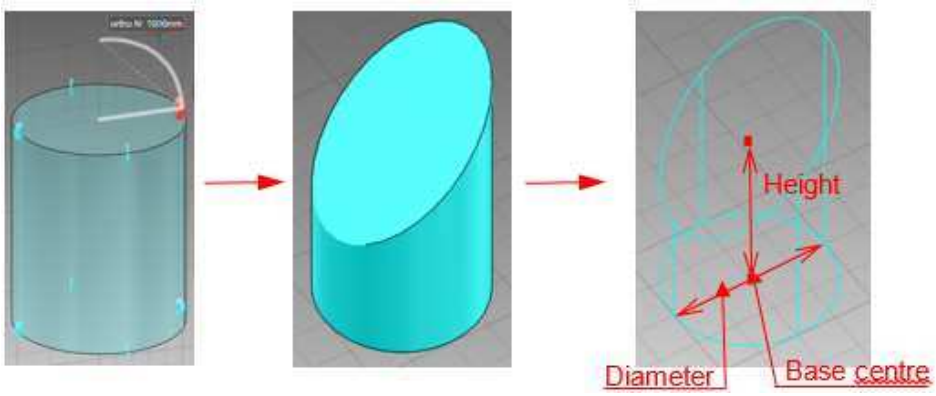
enter value for the height → **Enter**



- Slopped Cylinder Primitive

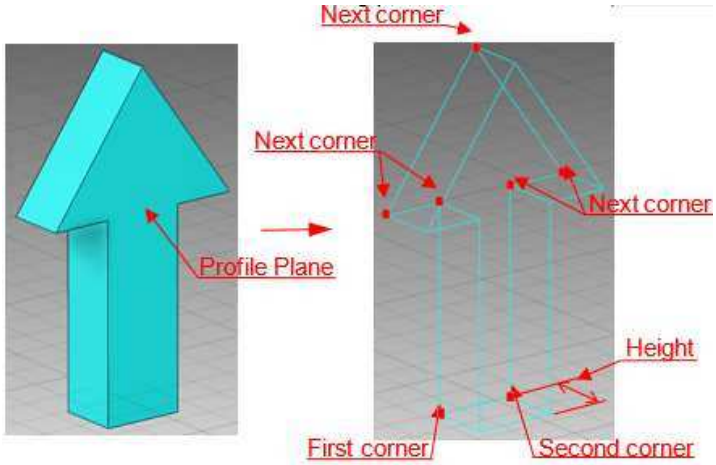


Click **Create Slopped Cylinder Primitive** → Specify the base centre or | Rel | E: 1000mm | N: 0mm | U: 0mm enter values in each axis for the base centre → Specify the diameter or | D: 2000 enter value for the diameter → Specify the height or | D: 2500 enter value for the height → Specify Rotation or <Finish> Move mouse to a curved arrow  and click → Specify Angle | A: 45 identify Angle for up or down Plane or both Planes → **Enter**



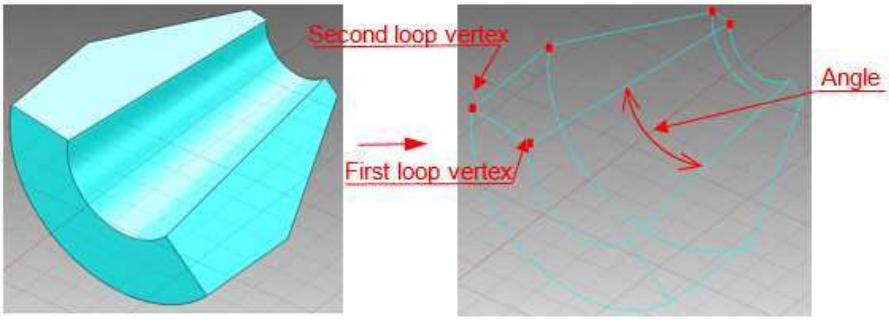
- Extrusion Primitive

Click **Create Extrusion Primitive** → Specify first corner (fillet radius: 0mm) or | Rel | E: 0mm | N: 0mm | U: 0mm insert for first corner → Specify second corner (fillet radius: 0mm) or | Rel | E: 0mm | N: 0mm | U: 1000 insert for the second corner → Specify next corner (fillet radius: 0mm) or | Rel | E: -300mm | N: 0mm | U: 1000mm insert for the next corner → Specify next corner (fillet radius: 0mm) or <Height> | Rel | E: 400mm | N: 0mm | U: 1700mm insert for the next corner → continue for next corner for finishing profile of extrusion → Specify the height or | D: 400 insert for the height → **Enter**



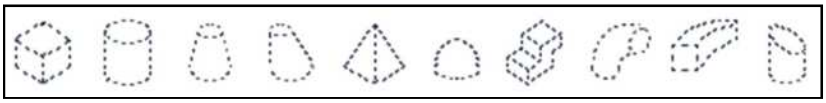
- Revolution Primitive

Click **Create Revolution Primitive** → Specify origin | Rel | E: 0mm | N: 0mm | U: 0mm insert for origin → Specify first loop vertex (fillet radius: 0mm) or | Rel | E: 0mm | N: 400mm | U: 0mm insert for first loop vertex → Specify second loop vertex (fillet radius: 0mm) or | Rel | E: 200mm | N: 500mm | U: 0mm insert for second vertex → Specify next loop vertex (fillet radius: 0mm) or | Rel | E: 200mm | N: 600mm | U: 0mm insert for next loop vertex → Specify next loop vertex (fillet radius: 0mm) or <Angle> | Rel | E: 0mm | N: 600mm | U: 0mm insert for next loop vertex → continue inserting for the next loop vertex → Specify revolution angle or | A: 145 insert for revolution angle → **Enter**



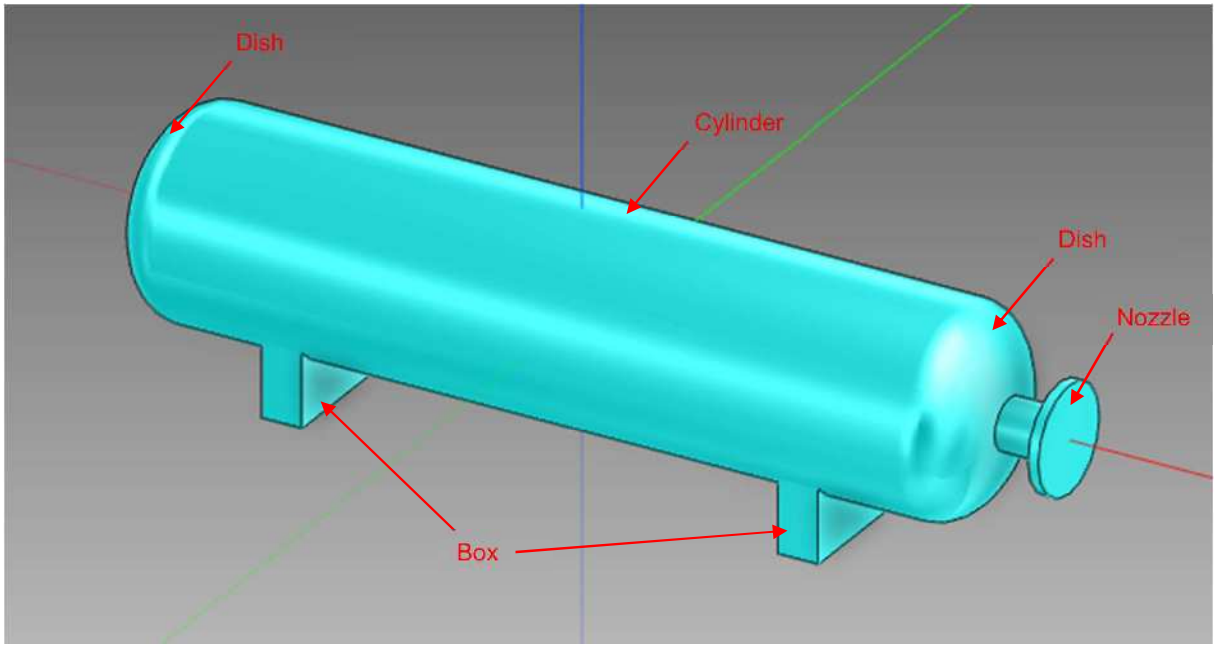
2.2.2.4 Negative Primitives

All primitives have an equivalent negative primitives (with the exception of a nozzle primitive) whose attributes are the same but have different names in the design hierarchy.



Use the principle and creating process of **Primitive** for **Negative Primitive** element

Sample of equipment:

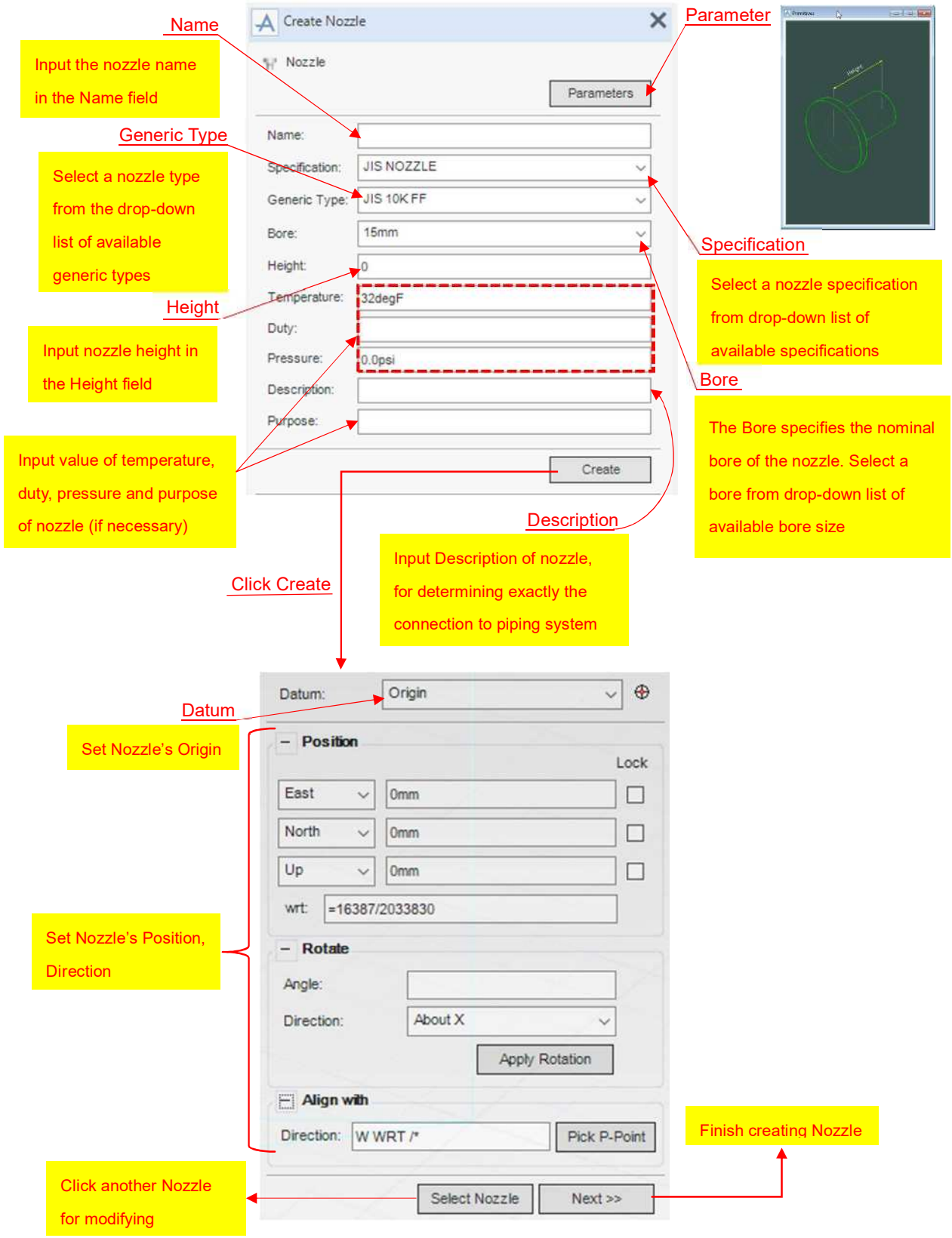


The simple storage vessel shown below can be constructed from a cylinder for main body, two dishes for the ends, two boxes for the support legs and a nozzle for the piping connection.

2.2.2.5 Nozzle

Nozzle is an interface point between the equipment and the connecting pipe. Nozzle is positioned, oriented and given a height dimension like other primitives; but must be selected form catalogue via a Specification

Use **Create Nozzle** window to define the nozzle specification

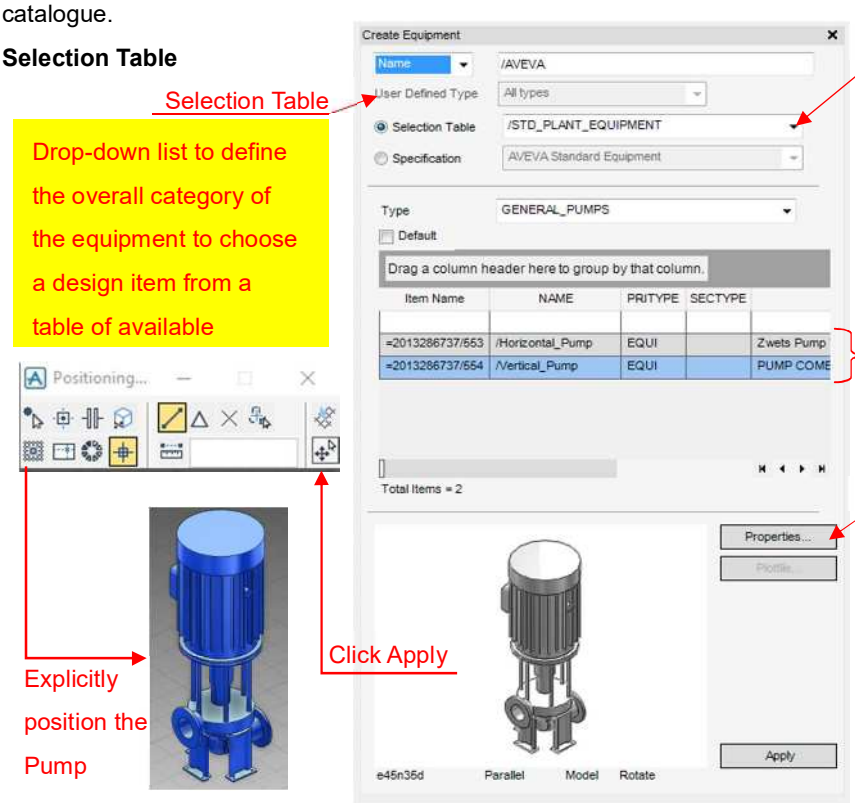


2.2.2.6 Equipment Standard

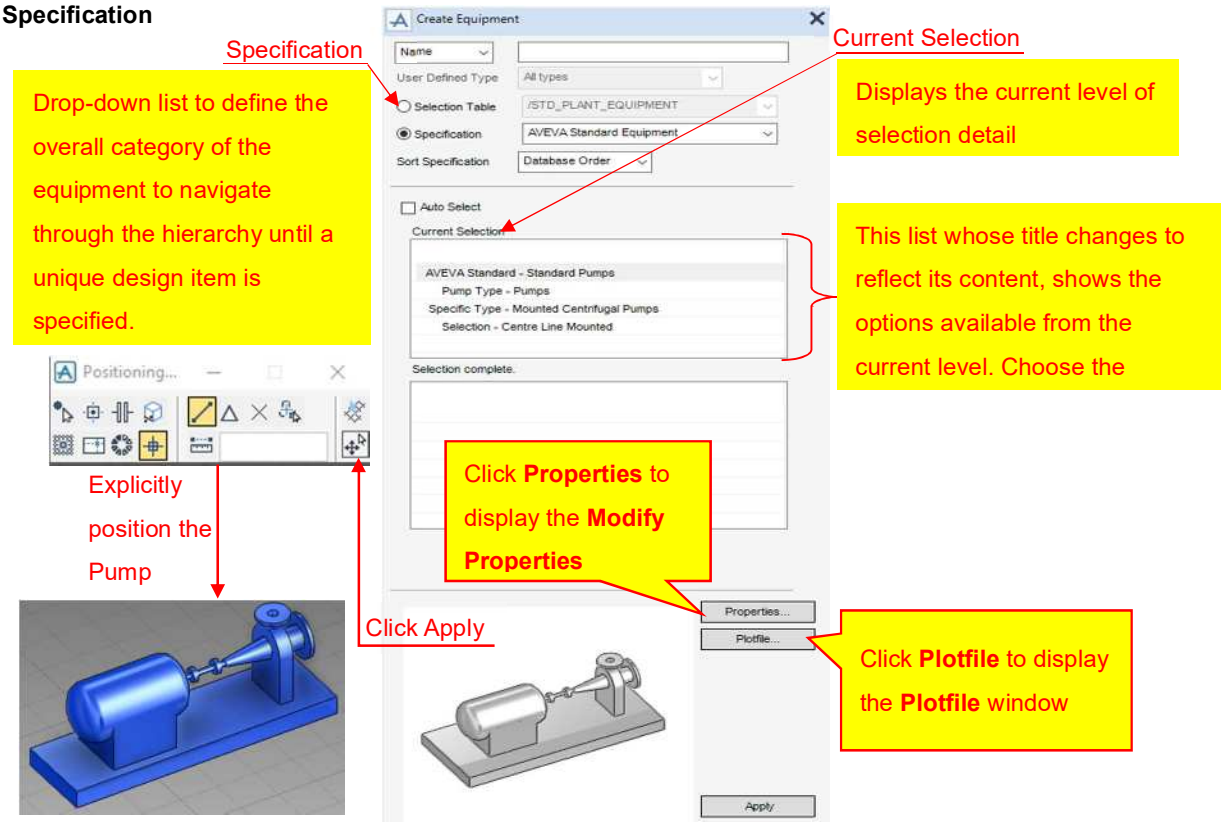
Use the **Standard Equipment** functionality to build up a parameterised equipment item (templates) based on a specification of standard equipment types.

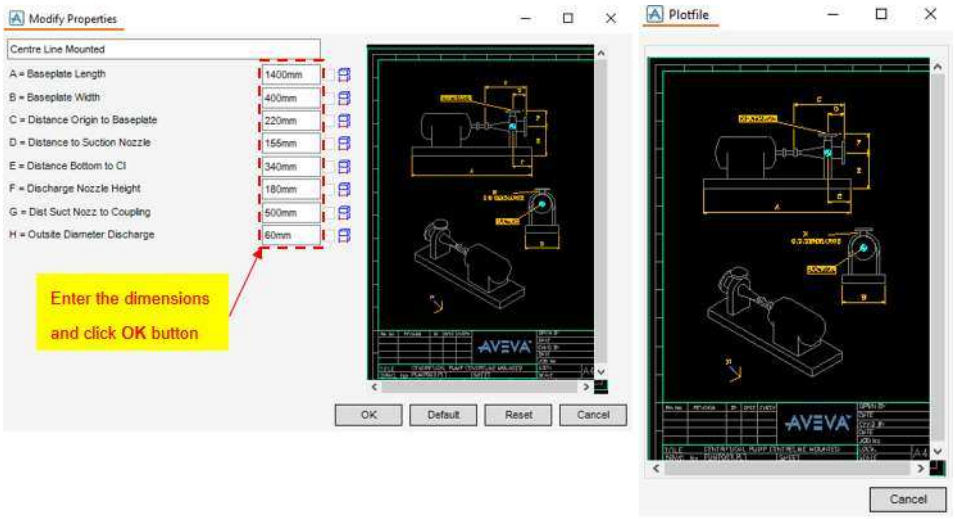
➢ Refer to **Design Templates** for further information about how to create and register template in catalogue module Standard equipment may be selected from a **Specification** or from a **Selection Table** as configured in the Equipment catalogue.

Selection Table



Specification





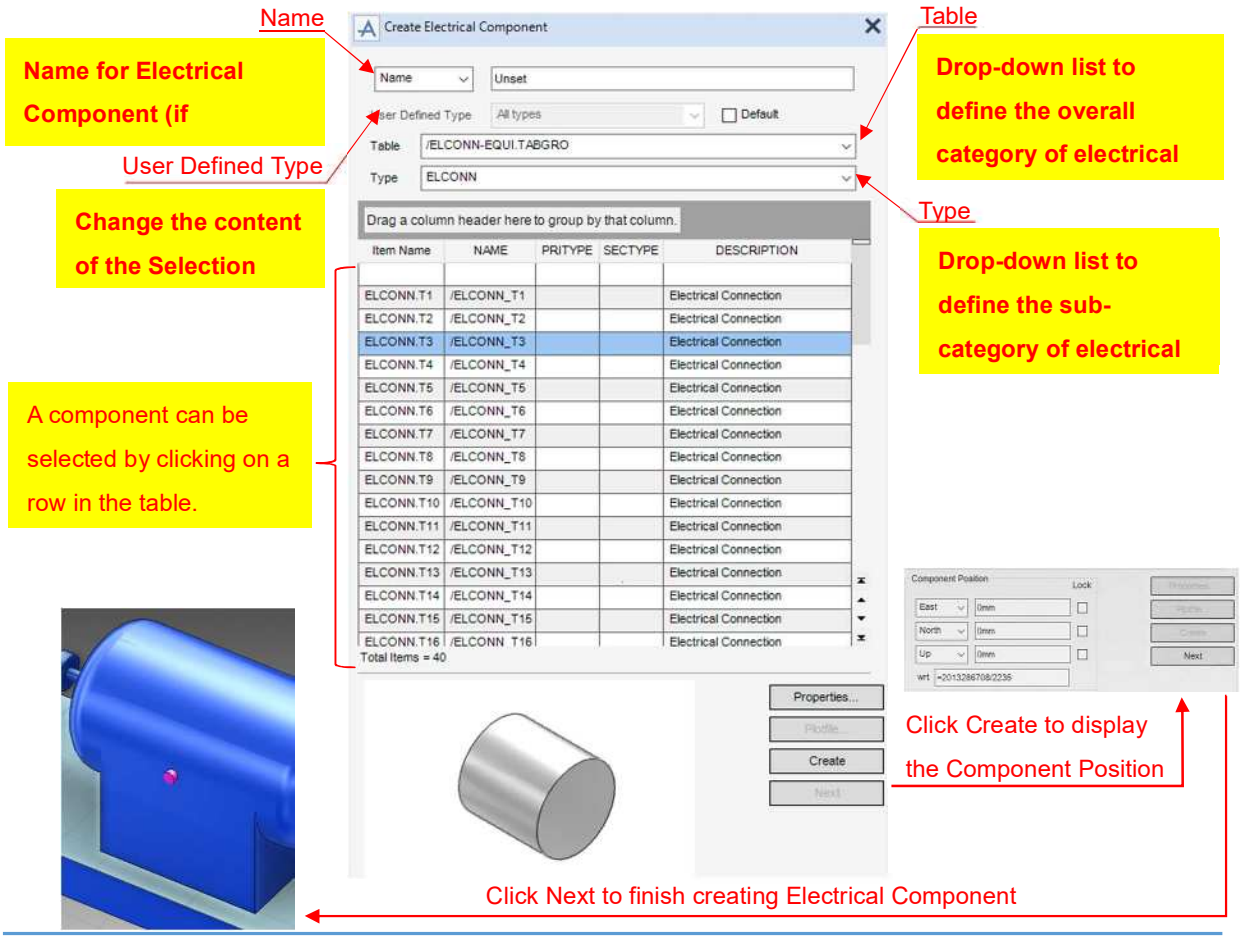
2.2.2.7 Electrical Component

An electrical component in equipment, is a connection point between equipment and cables and can represent anything from an individual cable gland to a piece of electrical equipment, such as generator.

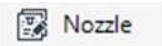
→ Create Equipment/ Sub-equipment (if necessary) in advance of creating electrical component

→ Use the **Create Electrical Component** window to select an electrical component from the list of available catalogue items.

→ If the project has been configured with User Defined Element Types based on electrical component element type, they display in the **User Defined Type** drop-down list.



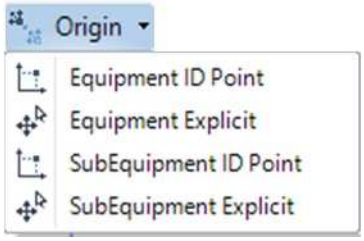
2.2.3 Modify group

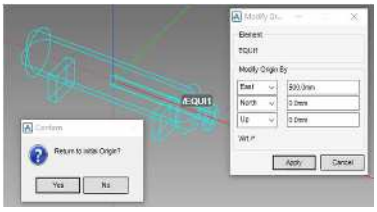
- Nozzle 

- Standard equipment 

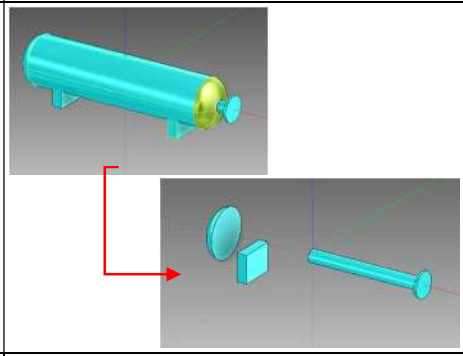
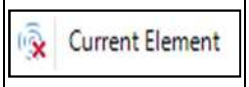
- Electrical componen 

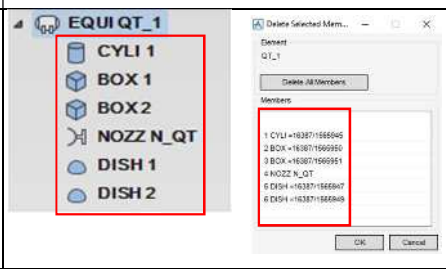
Same as each one's creating process

- Equipment/ Sub-equipment origin 

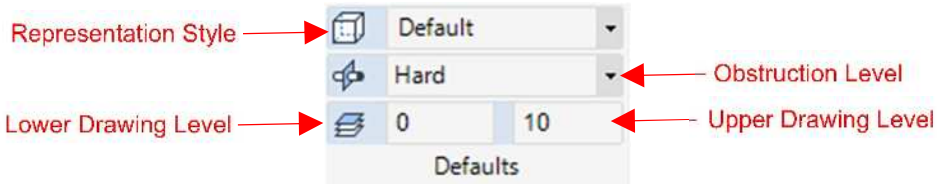
<u>Equipment/ Sub-equipment ID Point:</u> • Use **Cursor pick** in the **Graphical view**, the **Following prompt** is displayed. • Click the point used → click **No** to identify new origin or **Yes** to return to initial origin <u>Equipment/ Sub-equipment Explicit:</u> • Insert the values needed into **Modify Origin By/ Modify SUBE Origin** window. • Click **Apply** to modify or **Cancel** to discard any inputs → click **No** to identify new origin or **Yes** to return to initial origin.	<u>ID Point</u>*:*  <u>Explicit</u> 

2.2.4 Delete group

Function	Description	Image
	• Press **Ctrl + left click** one or more elements and click **Delete Selection/** press **Delete** button on keyboard → click **Yes** in **Confirm** message box • **Left click + Drag** elements that need to be deleted and click **Delete Selection/** press **Delete** button on keyboard → click **Yes** in **Confirm** message box	
	• Choose **acting element** and click **Current Element** deleting symbol → click **Yes** in **Confirm** message box	

	• Choose **acting element** and right click → click **Delete** → click **Yes** in **Confirm** message box	
CE Members	Click the **Current Element** and choose **CE Members** for deleting elements inside → click **OK**	

2.2.5 Equipment Default

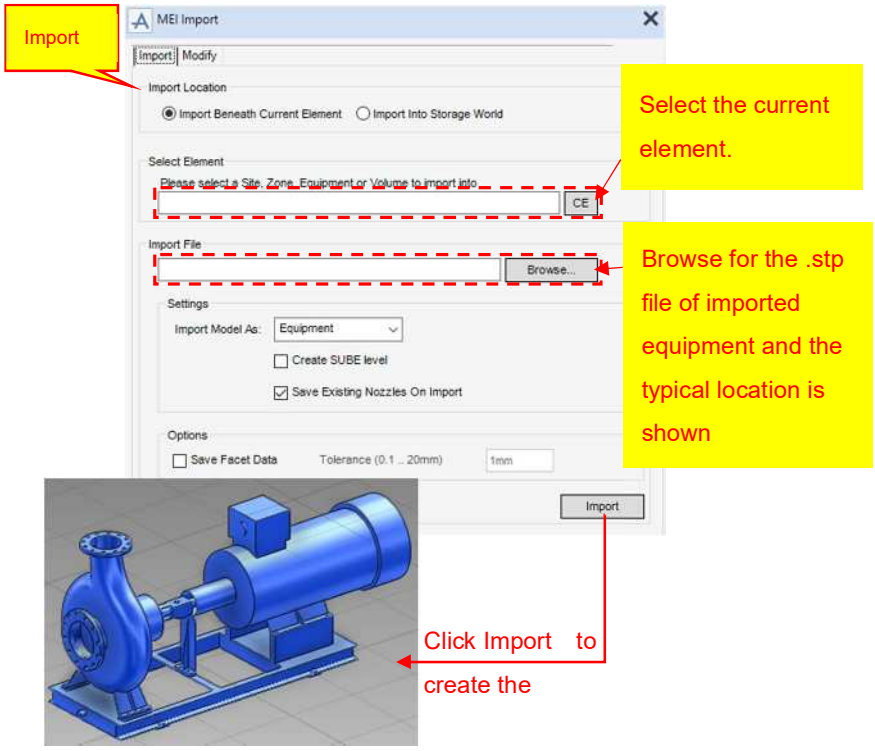


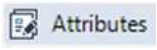
Each representation style corresponds to a given combination of these representation settings. References types can then reference such a type to change the appearance of the displayed component.

2.2.6 Importing Equipment

In the **Import** group of **TOOLS** Tab, click the MEI button; this displays the MEI Import form as below with **Import** Tab and **Modify** Tab

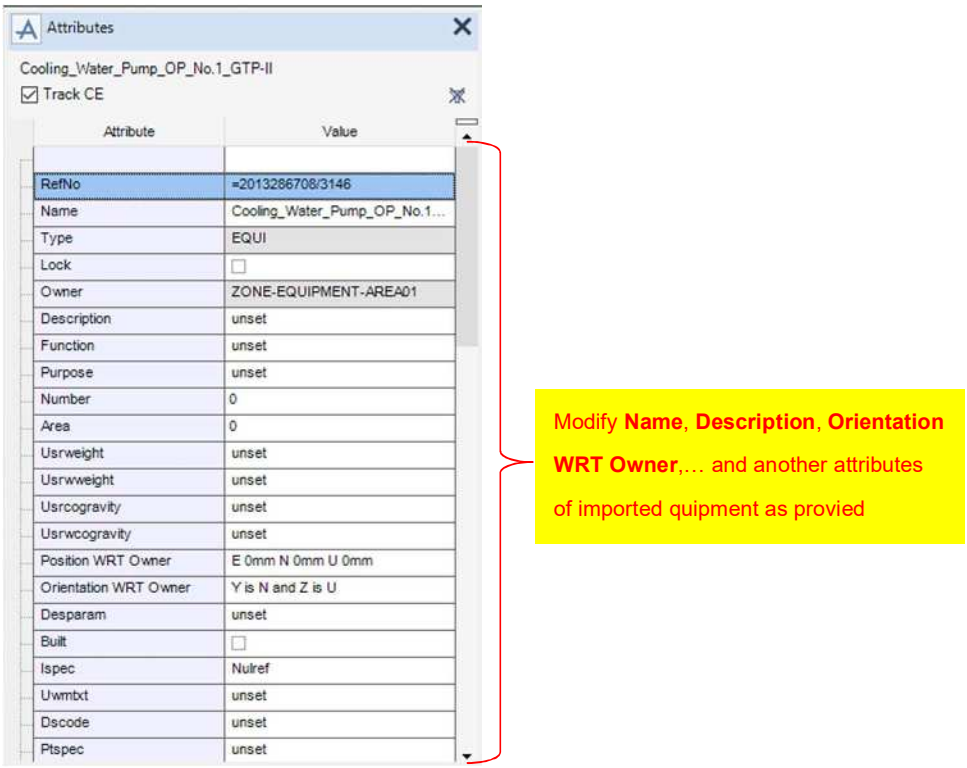
Open **Import** Tab



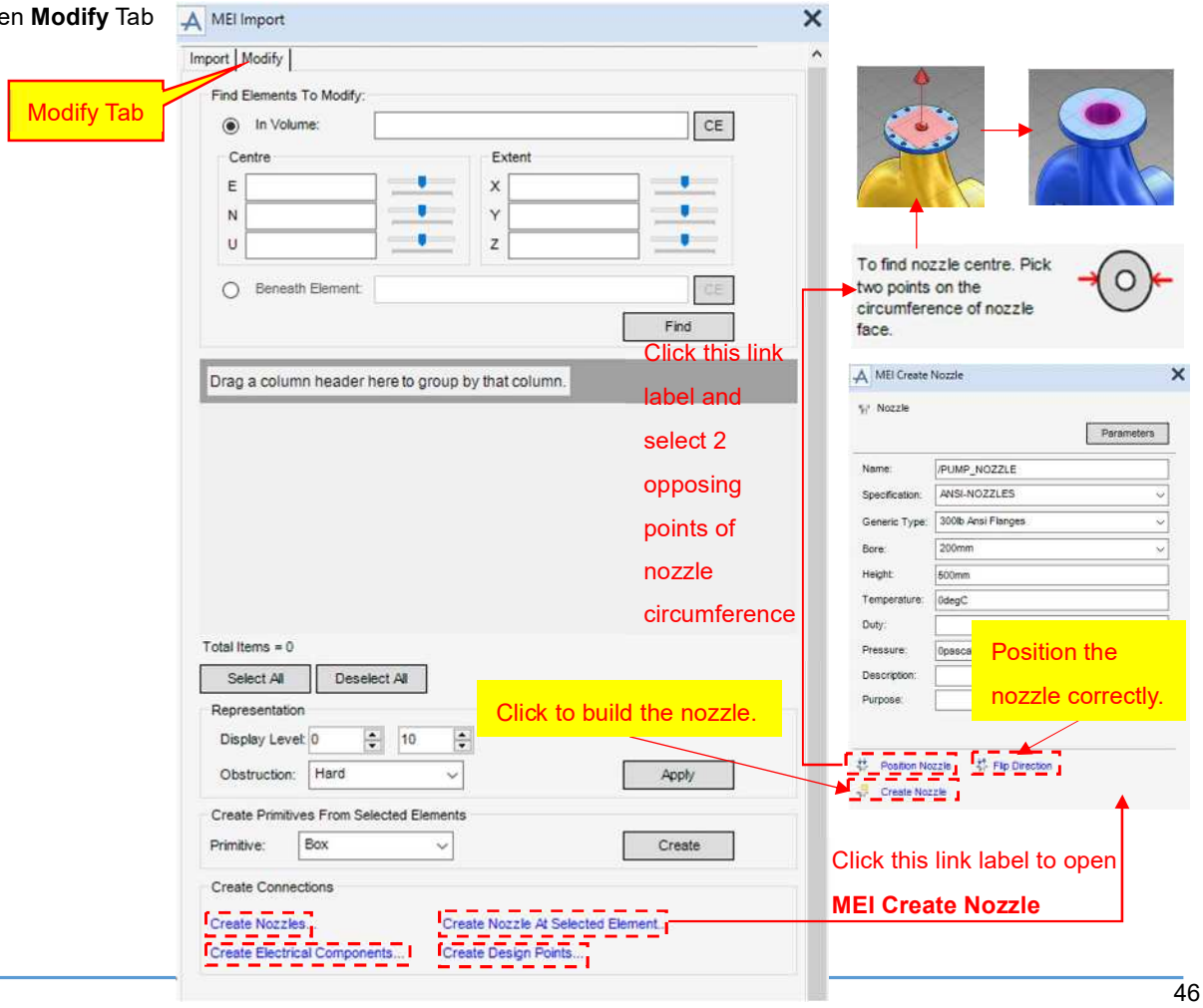
In the **Display** Group of **Home** Tab, click **Attributes** button 

to open **Attributes** form

→

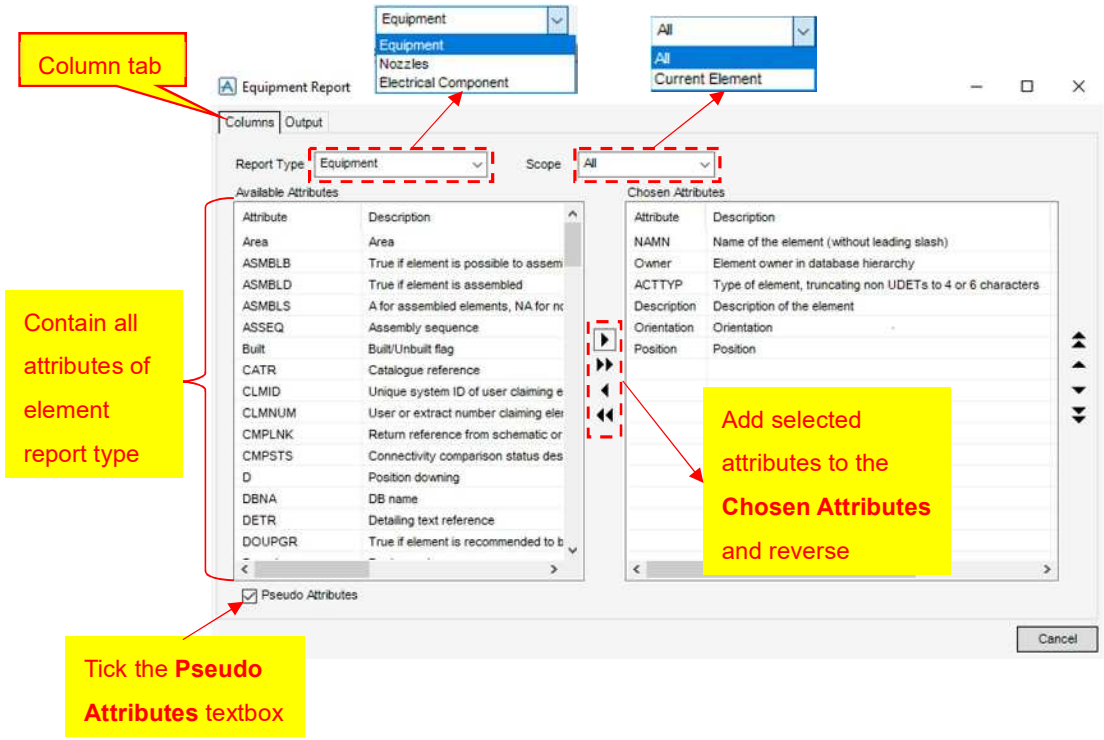


Open **Modify** Tab

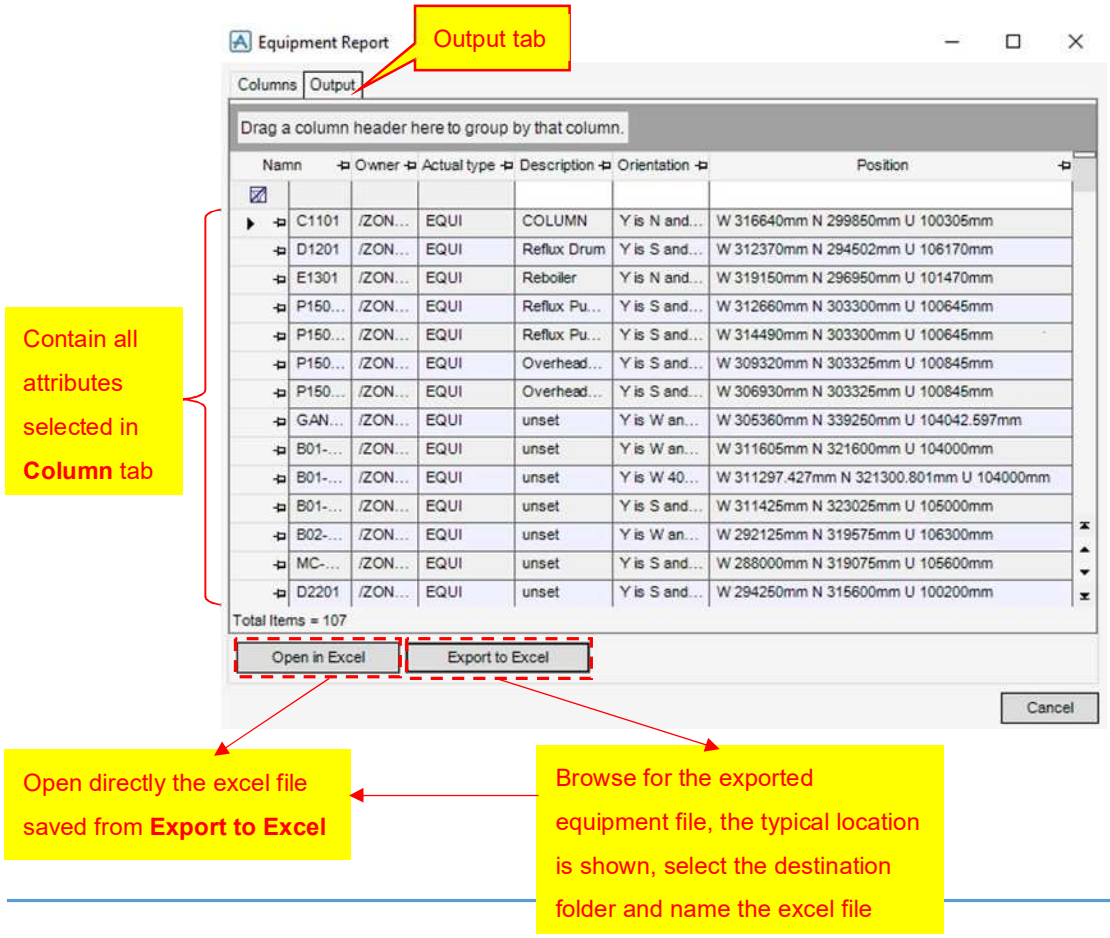


2.2.7 Equipment Report

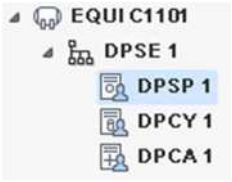
In the **Report** group of the **Equipment** Tab, click the **Equipment** button to open **Equipment Report** and go to **Column** tab.



Go to the Output section of **Equipment Report**



2.2.8 Tools

Design Points 

: Design Point Model Explorer

Design Point

Cartesiar Design Point

Create:

A Cartesian Design Point (DPCA), which is owned by a Design Pointset (DPSE), is specified in terms of its coordinates only

→ **Number:** Input a design point number into the **Number** field, select **Next Available** to display the next design point by default.

→ **Description:** Enter a description into the optional **Desciption** field.

→ **Function:** Enter a function into the optional **Function** field.

→ **Purpose:** Enter a purpose into the optional **Purpose** field.

→ **Connection:** Displays the **Point Connection** window

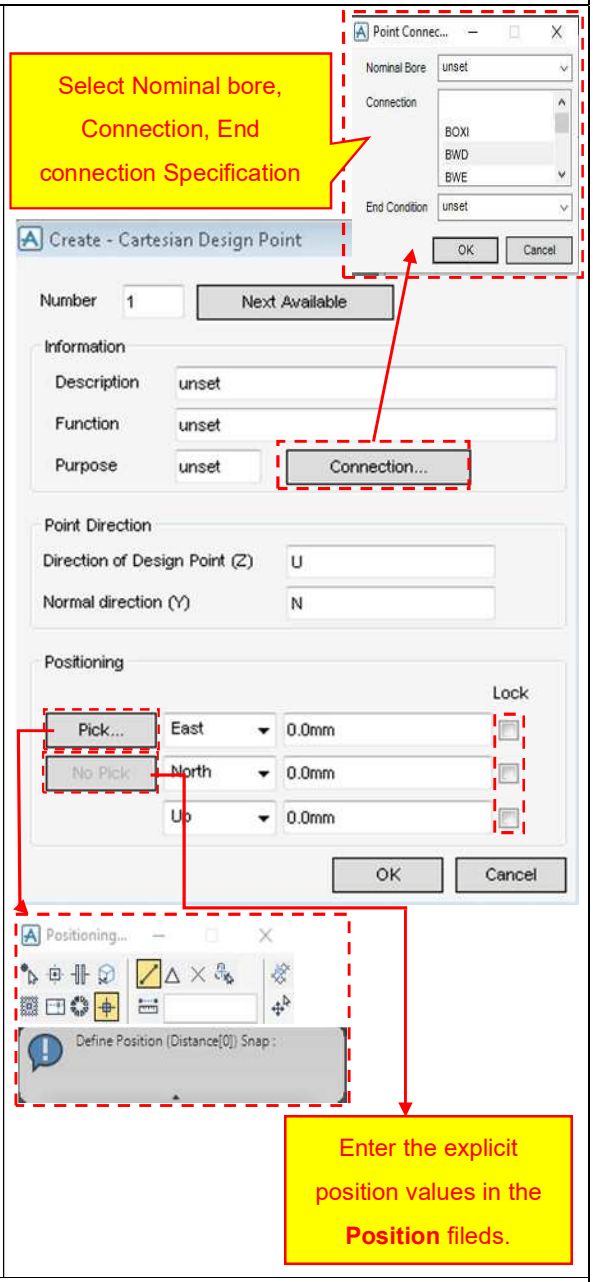
→ **Direction of Design Point(Z):** Enter a design point connection direction, ex: input **U** to direct the design point to face the U axes.

→ **Normal Direction (Y):** Enter a design point alignment direction, the Y direction is the rotation around the Z direction.

→ **Lock:** Select the check boxes to lock the explicit position of the design point.

Modify:

Use the **Modify Cartesian Design Point** window to modify the currently selected Cartesian Design Point. The functionality of the **Modify - Cartesian Design Point** is identical to that already described in **Create Cartesian Design Point.**



Cylindrical Design Point

Create:

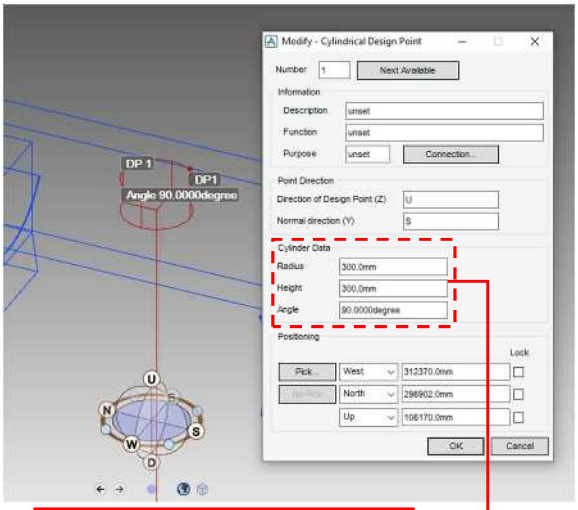
A Cylindrical Design Point (DPCY), is owned by a Design Pointset (DPSE) and is specified as a position on the surface of a cylinder at a given position and with given dimensions.

→ Function and operation is same as **Cartesian Design Point**, **Cylinder data** is the difference only to make the Design Point on the boundary of Cylinder according to values of **Radius, Height, Angle**

Modify:

Use the **Modify Cylindrical Design Point** window to modify the currently selected Cylindrical Design Point.

The functionality of the **Modify - Cylindrical Design Point** is identical to that already described in Create Cylindrical Design Point.



Enter a value to define the **Radius, Height** and **Angle** of the cylinder to position the design point.

Spherical Design Point

Create:

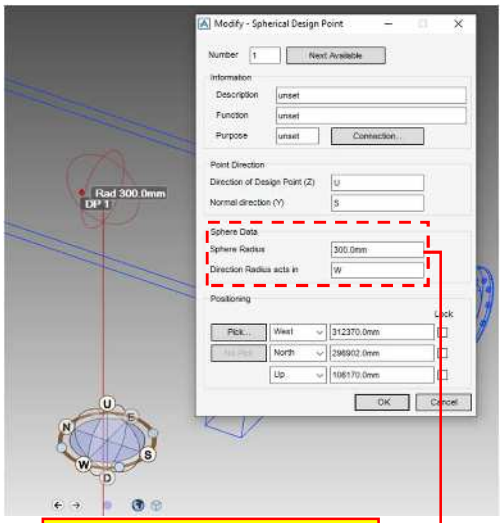
A Spherical Design Point (DPSP), is owned by a Design Pointset (DPSE) and is specified as a position on the surface of a sphere at a given position and with given dimensions.

→ Function and operation is same as **Cartesian Design Point**, **Sphere data** is the difference only to make the Design Point on the boundary of Sphere according to values of **Sphere Radius** and **Direction Radius acts in**

Modify:

Use the **Modify Spherical Design Point** window to modify the currently selected Spherical Design Point.

The functionality of the **Modify - Spherical Design Point** is identical to that already described in Create Spherical Design Point.

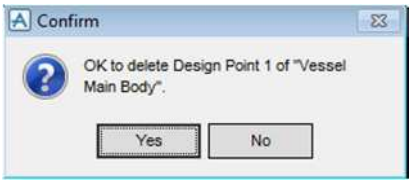


Enter a value to define the **Sphere Radius** and **Direction Radius acts in** of the sphere to position the design point.

Delete Design Point

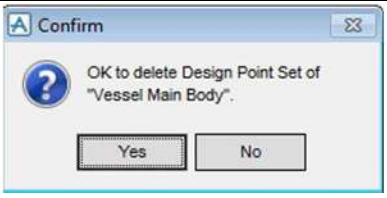
Delete Points:

Delete DPCA, DPCY, DPSP



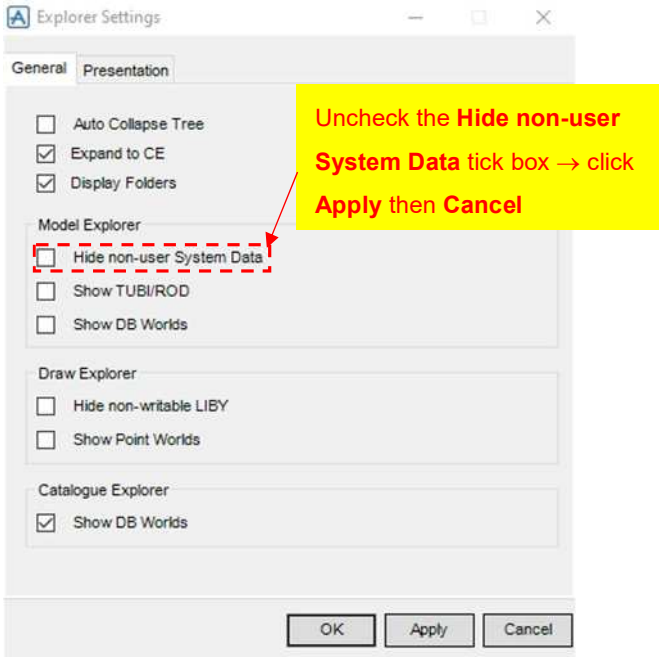
Delete Point Set:

Delete DPSE

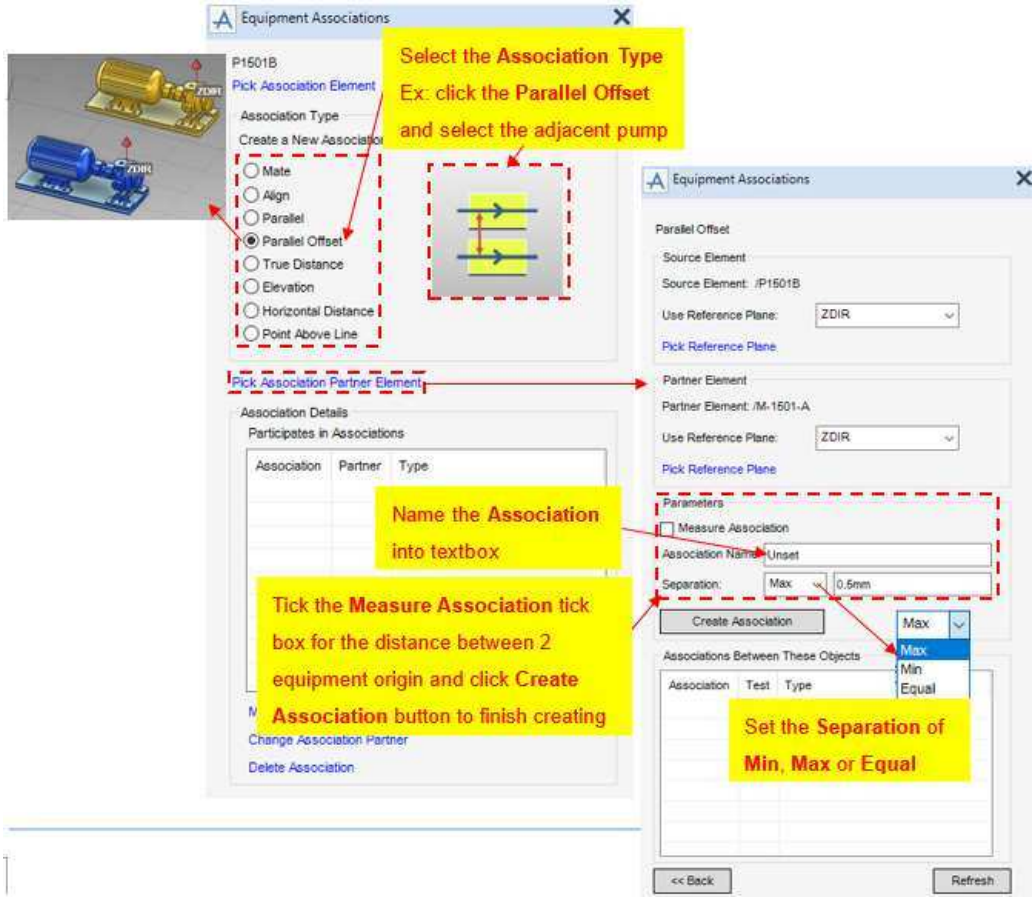


2.2.9 Association

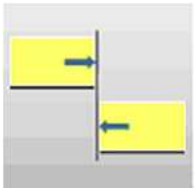
Open the **Project** tab, select **Options** then click **Explorer Settings** from **Configuration** tab to open the form below



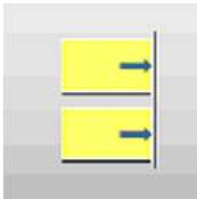
Select the equipment in a process equipment unit in **Model Editor** (Ex: pump unit) → click the **Equipment** button in the **Association** group of the **Equipment** tab to open the **Equipment Association** form.



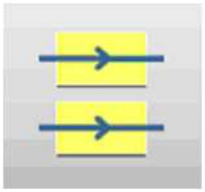
Use the **Association Type** to define the geometric restraint:



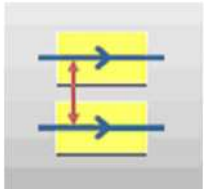
Mate: Two faces are touching (coplanar) or separated by a given distance. Directions defined by the tow face normal are parallel and in the opposite direction.



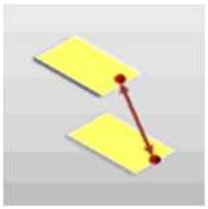
Align: Two faces are aligned (coplanar) or separated by a given distance. Directions defined by the two face normal are parallel and in the same direction



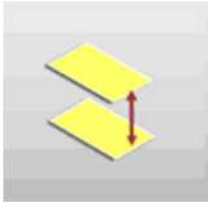
Parallel: Two directions are parallel, pointing in the same direction or in opposed directions



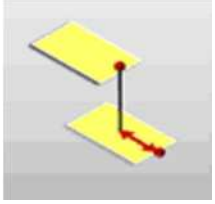
Parallel Offset: Two lines are parallel and separated by a given distance.



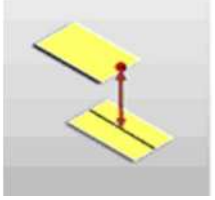
True Distance: Two points are coincident or separated by a given distance.



Elevation: The elevation of two points is separated by a given distance.



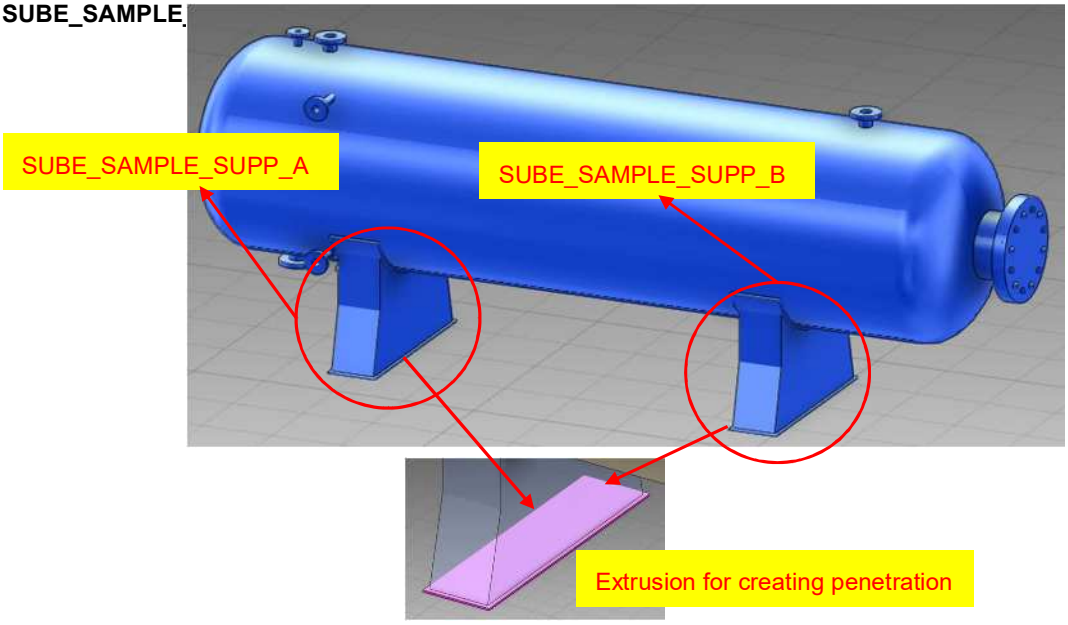
Horizontal Distance: One point is vertically above the other point, or separated in the horizontal plane by a given distance



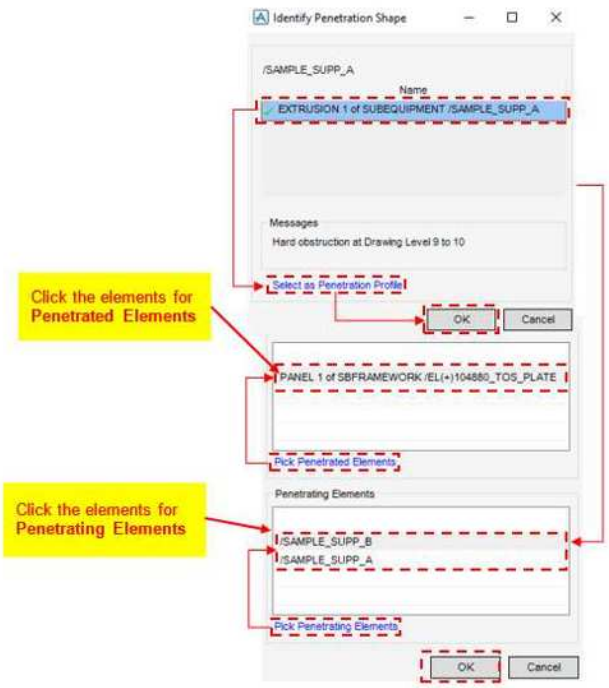
Point above line: A point is vertically above or below a line within a given tolerance.

2.2.10 Equipment Penetration

Add equipment **SAMPLE** with two sub-equipment name **SUBE_SAMPLE_SUPP_A** and **SUBE_SAMPLE_SUPP_B** to 3D view → In the **Defaults** group in **GENERAL** tab, set the **Default** to **Obstruction Volume** and **Soft** → Create 2 extrusions name at the base of 2 sub-equipment **SUBE_SAMPLE_SUPP_A** and **SUBE_SAMPLE**

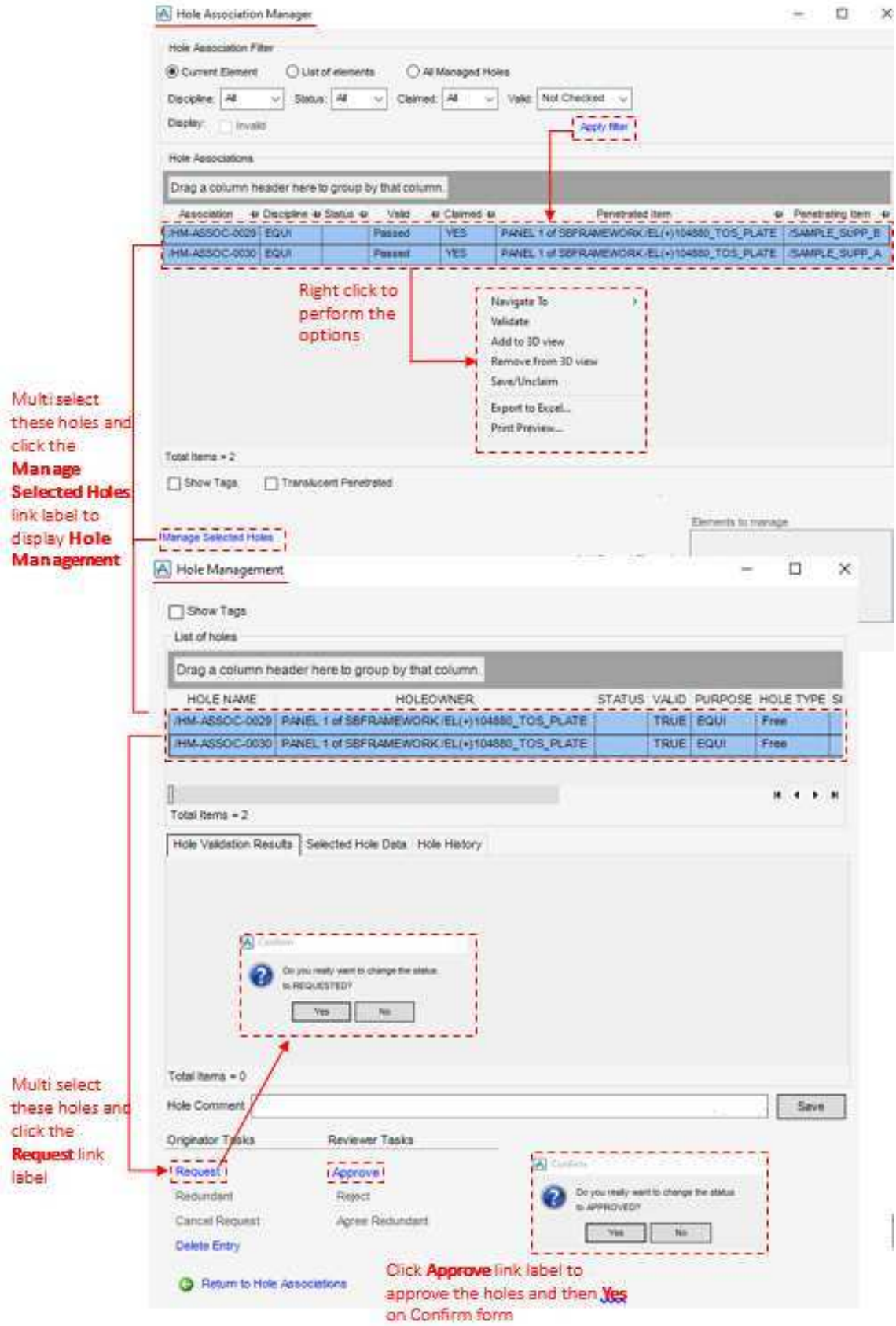


With the **SUBE** as the CE, click the **Penetration Profile** button in **Tools** tab → open **Identify Penetration Shape** window → select the extrusions → click **Select as Penetration Profile** → click **OK** button.



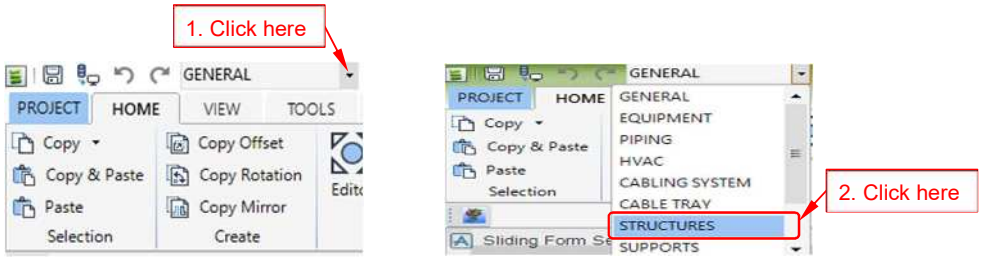
Use the **Create Penetration** form by clicking **Equipment** button in the **Penetration** group in **EQUIPMENT** tab, select the elements → click the **Pick Penetrated Elements** and **Pick Penetrating Elements** link label → click **OK** button

In the **Penetrate** group of **EQUIPMENT** tab, click **Holes** button → open **Hole Association Manager** form → make the structure the CE and click **Apply filter** linklabel → click the **Manage Selected Holes**



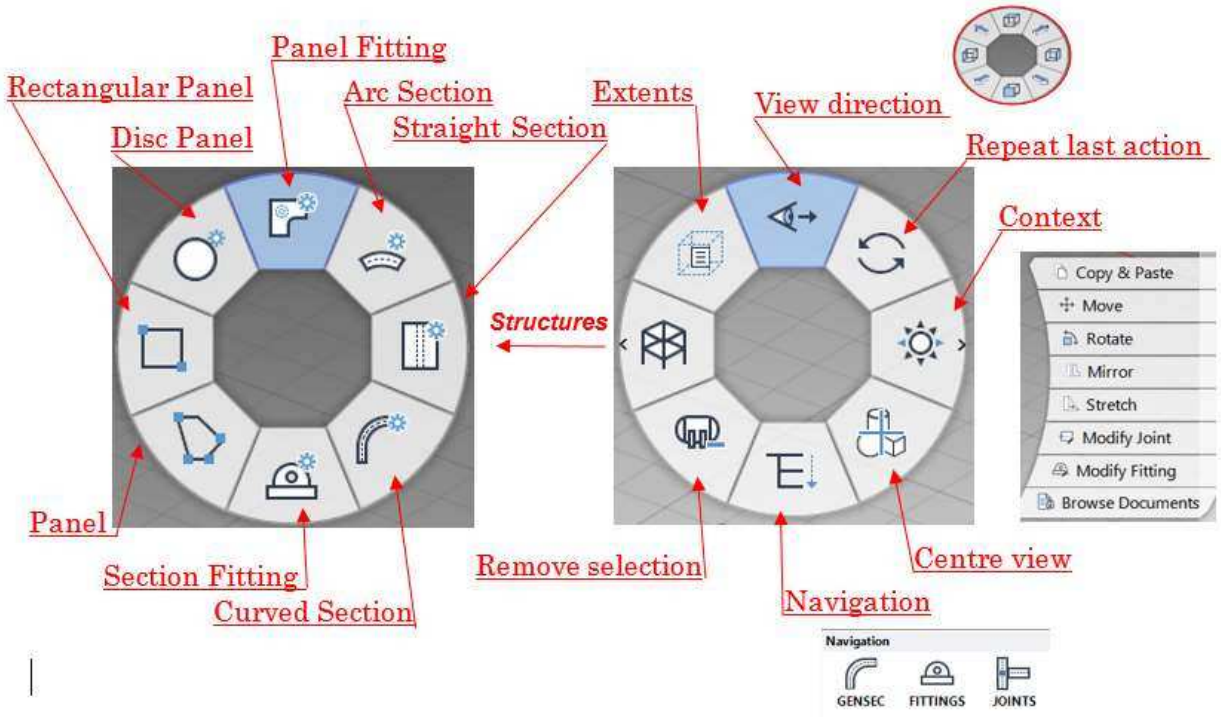
3. Structures

3.1 Open structures module



3.2 The interface of the structure model

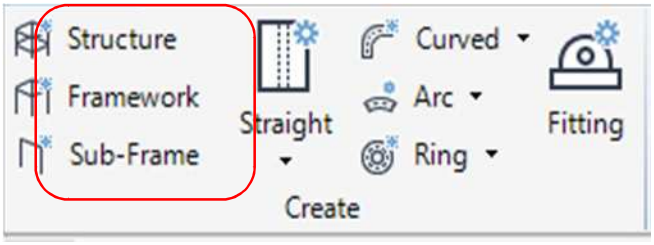
Navigator PowerWheel



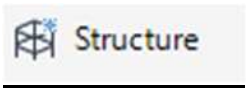
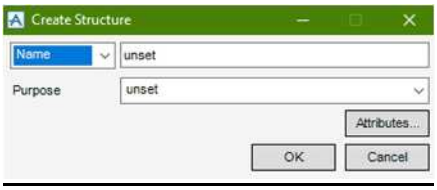
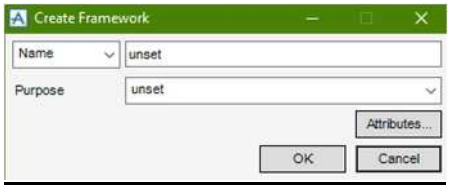
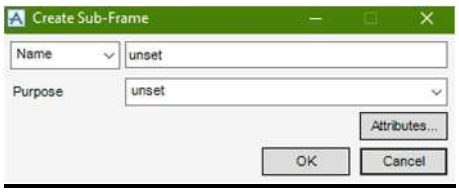
3.3 SECTIONS tab

Sections tab use to create, modify, connections and joint selection the GENSEC

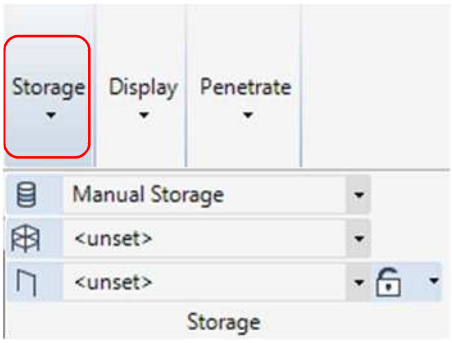
3.3.1 Create group from the SECTIONS tab



GENSEC may be created as straight, curved, arc or Ring configurations, with various creative methods for each type, available from the Create group of the SECTIONS tab.

	Tab **Structure** to create the new STRU of ZONE	
	Tab **Framework** to create the new FRMW of STRU	
	Tab **Sub-Frame** to create the new SBFR of FRMW	

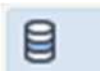
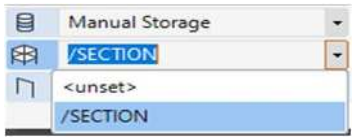
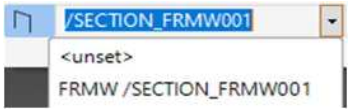
3.3.2 Storage from the SECTIONS tab



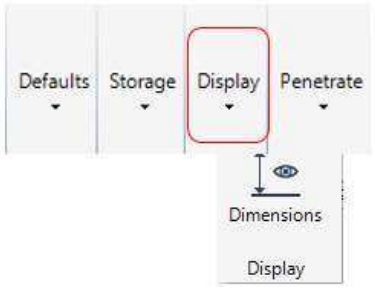
Stores structural sections in a specific area of the hierarchy.

There are two options for storing GENSEC and PANEL, Manual Storage and Grid Storage.

When Manual Storage is selected the user must navigate to the required FRMW or SBFR before modeling commences

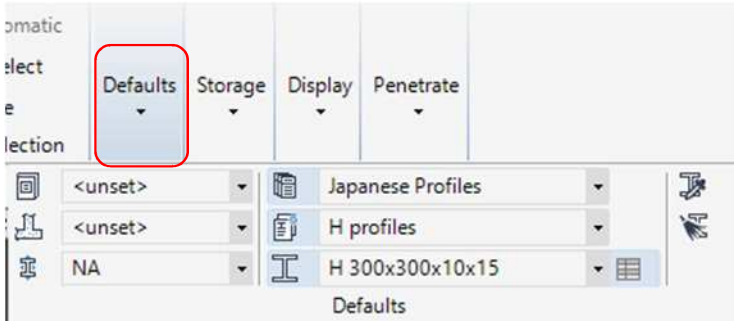
	Choose manual or Grid strorage	
	Choose area storage of the STRU to save	
	Choose area storage of the FRMW or SBFR to save	
	Choose button to lock or unlock storage	

3.3.3 Display from the SECTIONS tab



Display section dimension tags for the currently selected section elements in the 3D view. Repeat the command to remove the dimension tags from the 3d view

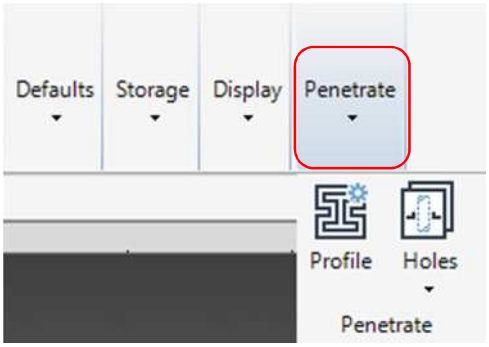
3.3.4 Defaults from the SECTIONS tab

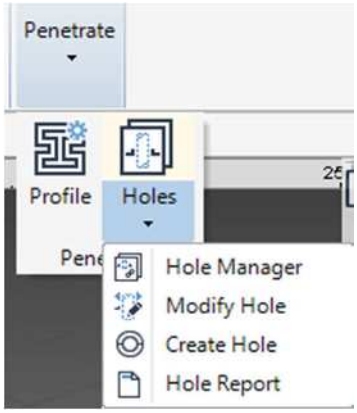


Select the Profile Specification, Generic Type and Profile from the options lists

	**Structural Classification:** Input a value or select the default structural classification from the drop-down list to be used for the creation of structural sections.
	**Default Material:** Input a value or select the default material from the drop-down list to be used for the creation of structural sections.
	**Default Justification:** Input a value or select the default justification from the drop-down list to be used for the creation of structural sections.
	**Profile Specification:** Input a value or select the default specification from the drop-down list to be used for the creation of structural sections.
	**Generic Type:** Input a value or select the default type from the drop-down list to be used for the creation of structural sections.
	**Profile:** Input a value or select the default profile from the drop-down list to be used for the creation of structural sections.
	**Copy Defaults GENSEC:** When you choose one GENSEC and you want to set profiles like this GENSEC you click this button
	**Modify Profile GENSEC:** You choose GENSEC you want to modify after that you choose generic type and profiles and then you click this button

3.3.5 Penetrate from the SECTIONS tab

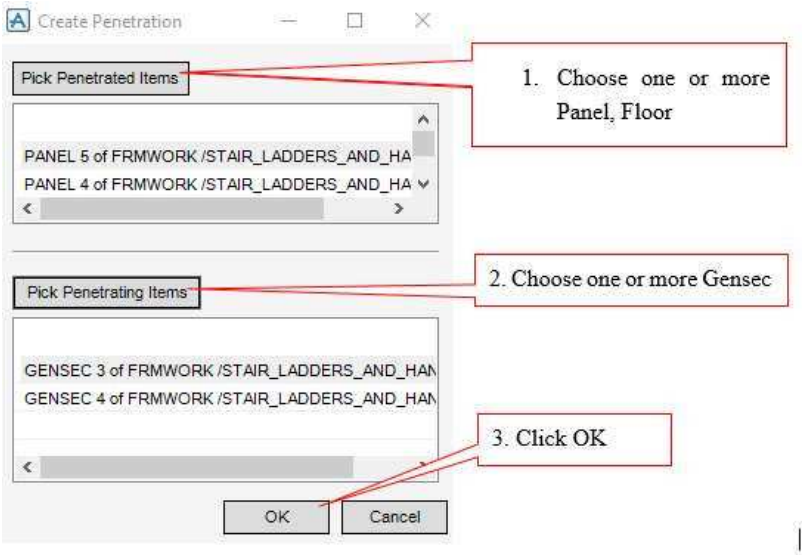




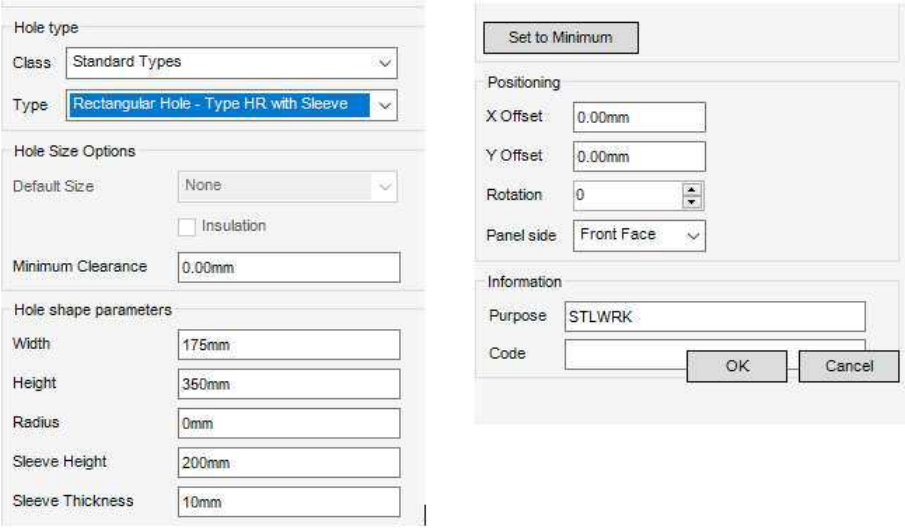
3.3.5.1 Profile tab

Create Penetration with Profile

Step 1: Click the button Profile to show table below



Step 2: After click OK it will show the table below



Class and type: to choose the type of penetration

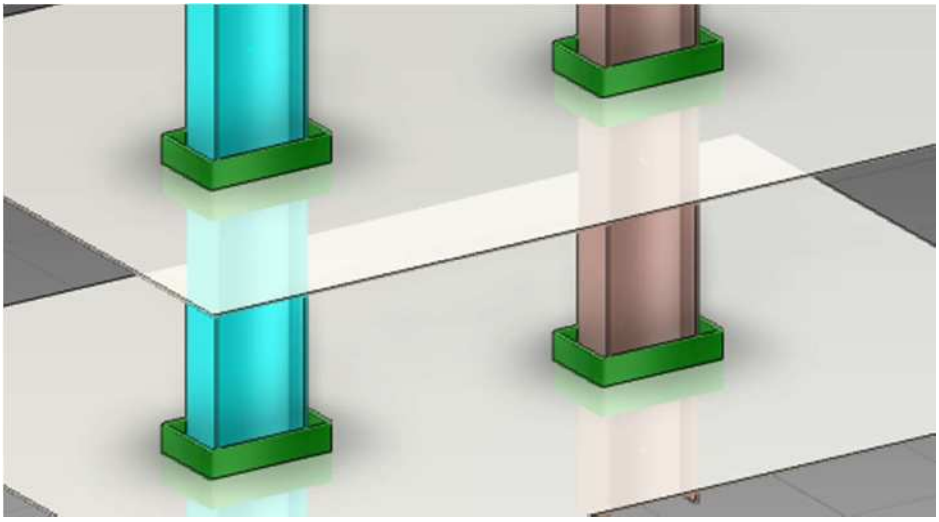
Minimum clearance: to set the minimum dimension with penetration and GENSEC. If you set 50mm so when you click button Set to Minimum the dimension with penetration and GENSEC will be 50mm

Hole shape parameters: to set dimension of penetration

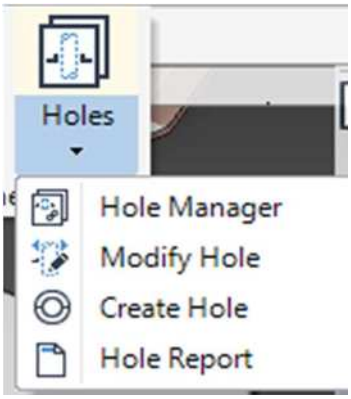
Positioning: to set position penetration with the GENSEC

Panel side: to choose front face or back face of the panel

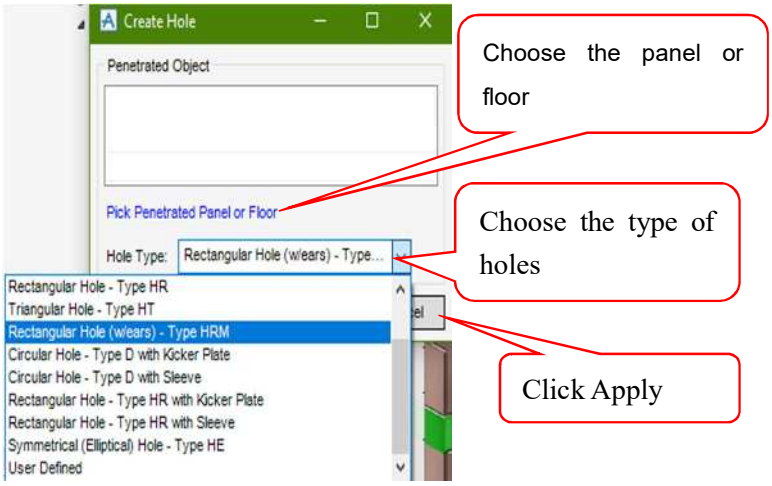
Then click OK to finish to create penetration with profile



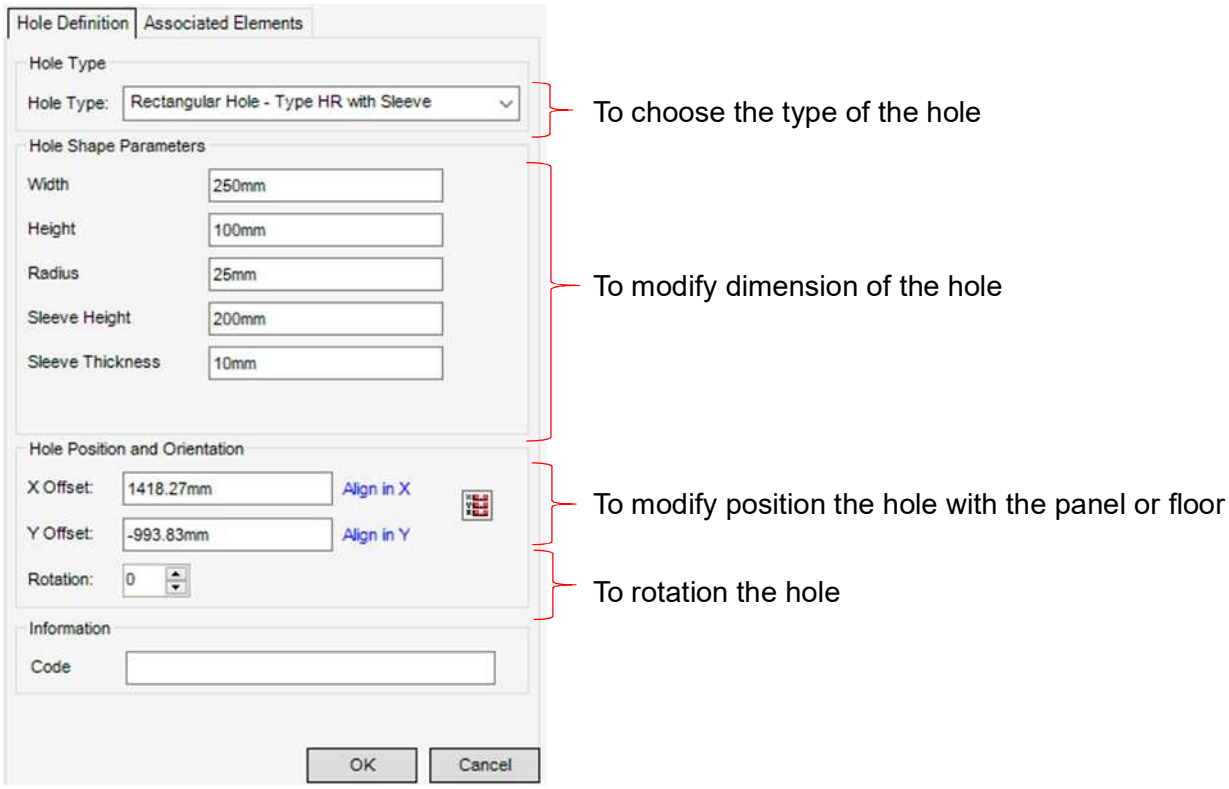
3.3.5.2 Holes tab



Create Hole: to create holes on the panel or floor



After click Apply it will show the table below

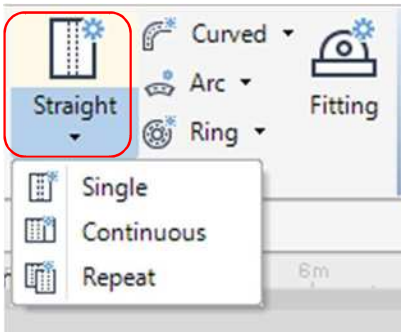


Modify Hole : to modify the type, parameter, position or rotation of the hole

Hole Manager and hole report : to filter find all the holes on project

3.3.6 Create GENSEC from the SECTIONS tab

3.3.6.1 Straight GENSEC



The Straight button has the following option:

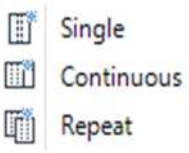
Single: Creates one straight GENSEC.

Continuous: Creates any number of GENSECs with the end position of the first being the start position of the second, etc.

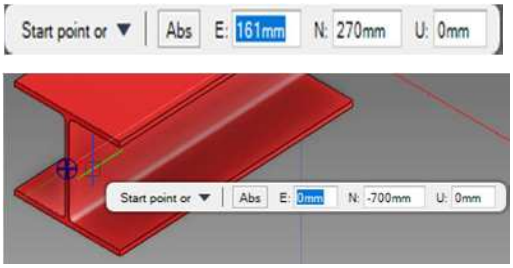
Repeat: **C**reates any number of GENSECs using Single the mode by repeating the command.

How to create:

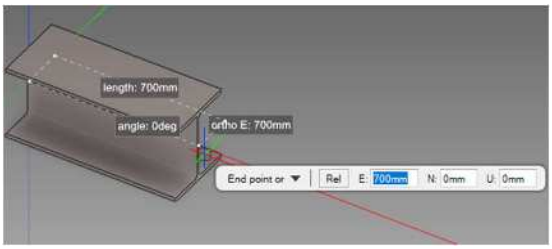
Step 1: Click one of three button



Step 2: Choose the xyz coordinates start point or choose pick point

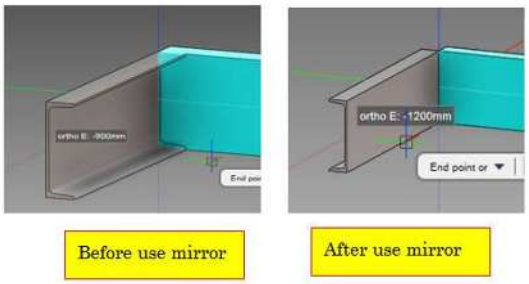


Step 3: Choose the xyz coordinates end point or choose pick point.



End point button has the option and pressing the down cursor key displays the options below the prompt

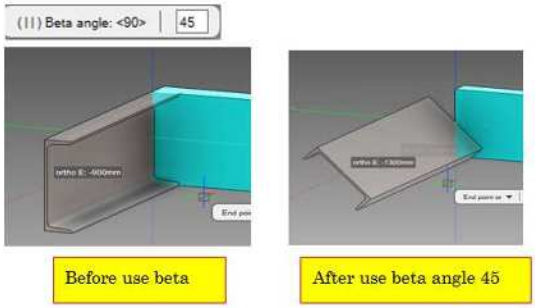
Mirror: Press the down arrow key and select Mirror to reposition the section as a reflection of the original position.



Beta: Press the down arrow key and select Beta to rotate the section.

You are prompted to input Beta angle <#>

Input an angular value in the field and press Enter to set the angle of the section.

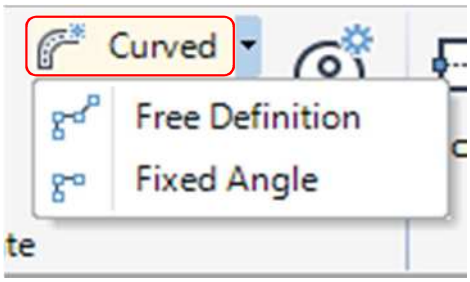


Parallel: Press the down arrow key and select Parallel to orientate the web of the section parallel to the X, Y plane

Perpendicular: Press the down arrow key and select Perpendicular to orientate the web of the section perpendicular to the X, Y plane.

Back: Press the down arrow key and select Back to undo the last action.

3.3.6.2 Curved GENSEC



The **Curved** button has the following options:

Free Definition: Creates a curved GENSEC with as many points as required and variable curve angles

Fixed Angle: Creates a curved GENSEC with as many points as required and the same fixed curve angle

End point button has the option and pressing the down cursor key displays the options below the prompt

Radius: Press the down arrow key and select Radius to modify the radii of the curved section. You are prompted to Curve radius <#>

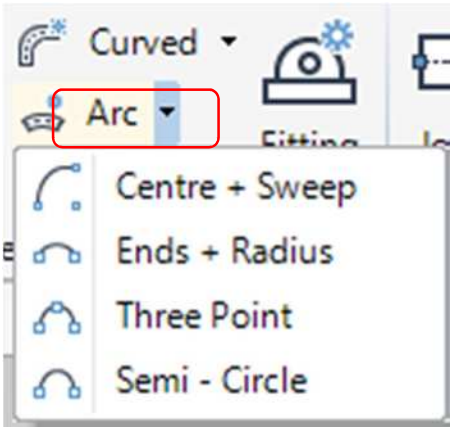
Input a dimensional value in the field and press Enter to set the radii of the curved section.

Angle: Press the down arrow key and select Angle to create a curved section with a specified angle. You are prompted to Curve angle <#>

Input an angular value in the field and press Enter to set the angle of the curved section.

Mirror, Beta, **Parallel, Perpendicular and Back:** The same the straight GENSEC

3.3.6.3 Arc GENSEC



The **Arc** button has the following options:

Center + Sweep: Requires a center point, diameter (or radius), start angle and sweep angle

Ends + Radius: Requires a start point, endpoint, and a point to define the arc direction

Three Point: Requires three points through which the arc passes

Semi-Circle: Requires a start and end point and a point to define the arc direction

3.3.6.4 Ring GENSEC

The **Ring** button has the following options:

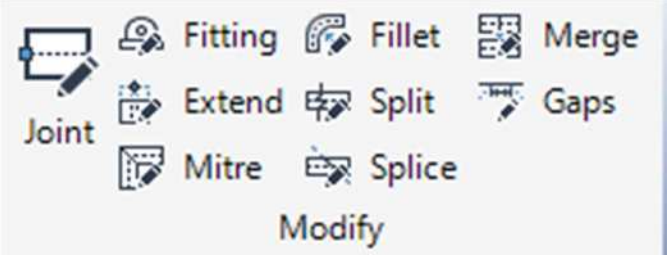
Centre +: Requires a center point and a diameter (or radius)

Three Point: Requires three points through which the arc passes

Two Point: Requires two points through which the arc passes. The second point defines the ring diameter

3.3.7 Modify GENSEC from the SECTIONS tab

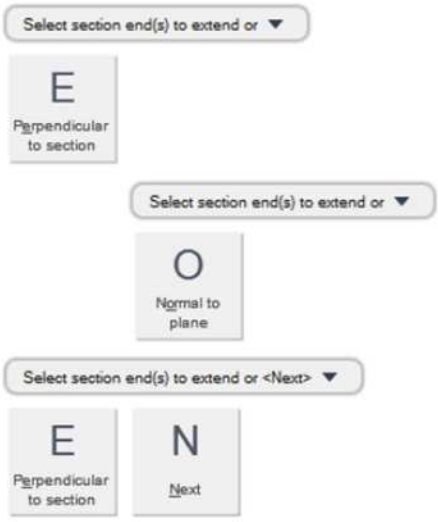
3.3.7.1 Extend button



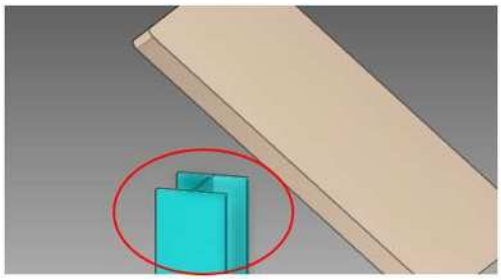
Lengthens or shortens sections using a plane created from a maximum of three identified points, a panel intersection, a local coordinate system intersection or an offset.

First: Choose button **Extend** and then choose the GENSEC or more GENSEC you want to extend.

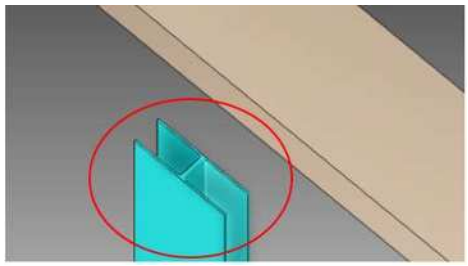
Second: Press the down arrow key to choose more options.



Perpendicular: To extend a section end to align with the plane without cutting the endpoint.



Normal to plane: To extend a section end to align with the plane to cut the endpoint.



Third: After you choose **Perpendicular** or **Normal to plane** you click button **Next** and choose more options below.

Panel: To extend the GENSEC to a panel.

Lcs: To extend a section end to the origin of the local coordinate system.

Offset: To create a plane that is offset from the picked points. You are prompted to specify offset Input a dimensional value in the field and press Enter to define the offset value.

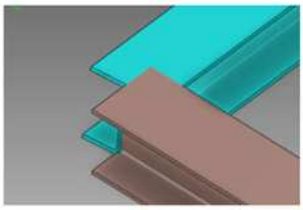
The last: After you click button **Next** and you don't want to choose **Panel**, **Lsc** or **Offset** you can choose three-point from XYZ coordinates point or choose pick point to make a plane to extend

3.3.7.2 Mitre button

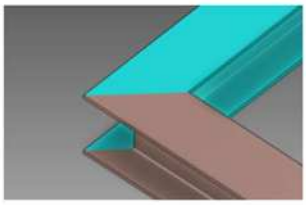
To extend straight section end to another straight section end. The Start Cut and End Cut properties of each section is modified to indicate the cut angle of the mitre end.

First: Choose button **Mitre**

Second: Click two GENSEC you want to mitre



Before use mitre



After use mitre

3.3.7.3 Fillet button

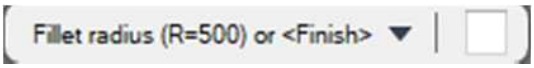
Fillet to merges two non-parallel to be a curved fillet. You are prompted to pick first GESNSEC to fillet and click to identify the second GESNSEC apply a fillet.

You are then prompted to Fillet radius (R = #) or <Finish>. The default is 1000mm.

Input F key and press Enter to end the command.

First: Choose button **Fillet**

Second: Click two GENSEC you want to fillet after you choose two GENSEC you must to input radius



Before use fillet

After use fillet

3.3.7.4 Split button

Splits a GENSEC to be two or multiple GENSEC

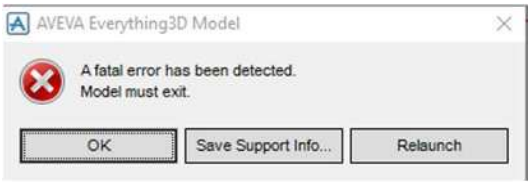
First: Choose button **Split**

Second: Click the GENSEC or more GENSEC you want to split and then you press the down arrow key and select Next after that you have two options to split.

Option 1: Choose one GENSEC, PANEL or a lot of GENSEC, PANEL to split and then press the down arrow key and select Next to finish or press key Enter to finish.

Option 2: You don't want to choose GENSEC you press the down arrow key and select Plane and then you choose three-point to make the plane to split.

Note: *with option 1 you can choose a lot of GENSEC or PANEL to split but you cannot choose GENSEC and PANEL together => E3D will be error and have a tab to a notification to exit. Option 2 you just choose only three-point to make plane to split.*

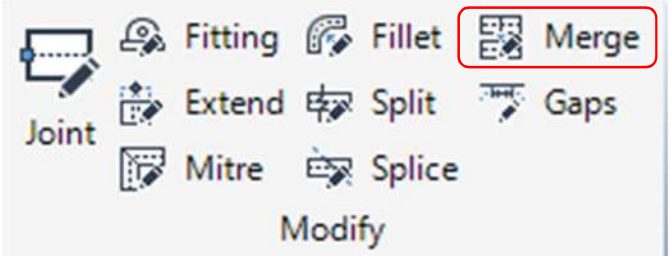


3.3.7.5 Splice button

Splice divides GENSEC into smaller lengths using a splice

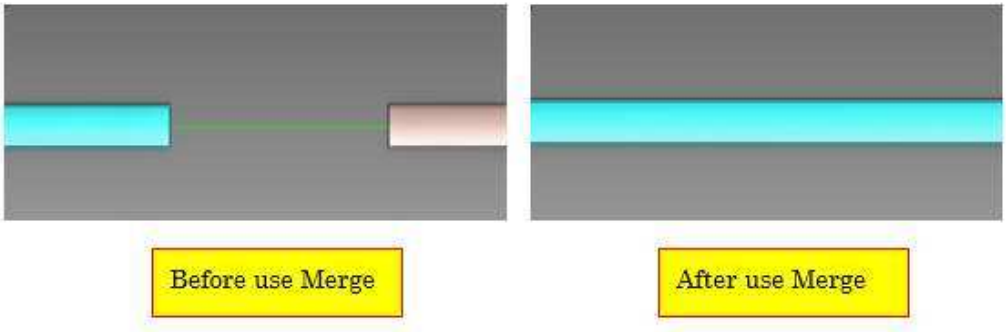
The way use this button **Splice** is the same button **split** but something is different like below**.**
With option 1 you just choose only one PANEL to splice and you can not choose GENSEC to splice.

3.3.7.6 Merge button

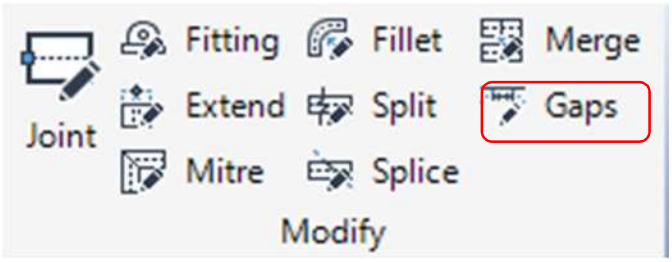


Merges two GENSEC to be one GENSEC

First: Choose button **Merge**
Second: Click the first GENSEC after that choose second GENSEC to merge

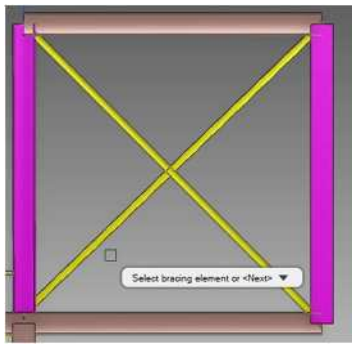


3.3.7.7 Merge Gaps

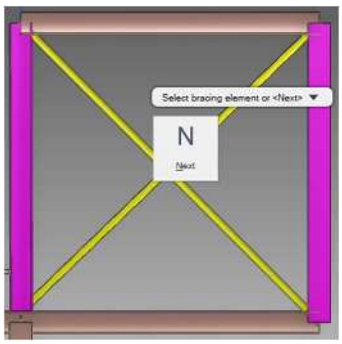


Creates a standard bracing configuration. A bracing gap can be created to define the gap between the bracing connection and a reference element.

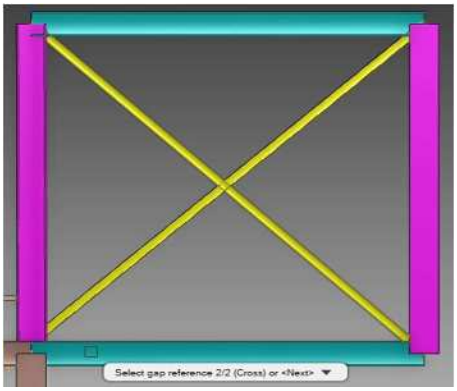
First: Choose button **Gaps**

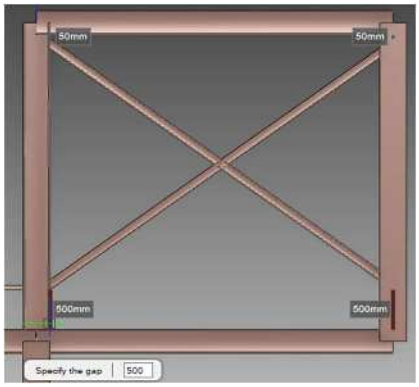


Second: Press the down arrow key and select Next or press key N on your keyboard and Enter.

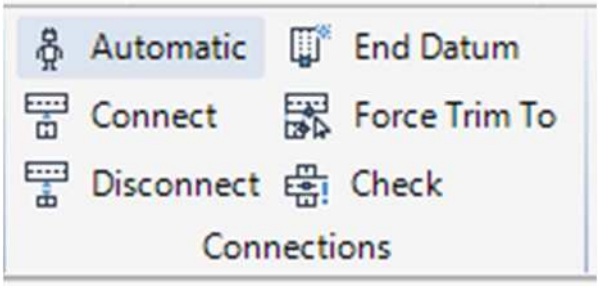


Third: Selection bracing element and then you click gaps to modify distance and press Enter to finish.



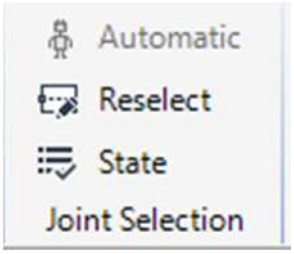


3.3.8 Connections group from the SECTIONS tab



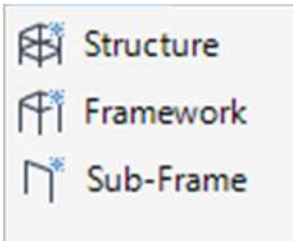
Automatic	Sets the default connection status when a new section is created. If the selection is highlighted, the automatic section end connection is on.
Connect	Connects two or more GENSEC together
Disconnect	Disconnects two or more GENSEC
End Datum	
Force Trim To	
Check	

3.3.9 Joint Selection group from the SECTIONS tab



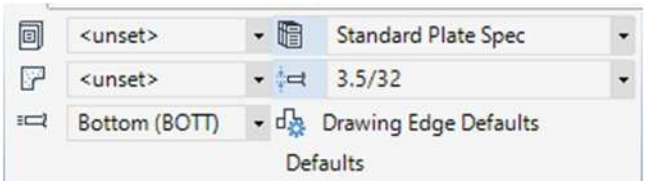
3.4 PLATES tab

3.4.1 Create group of the PLATES tab.



The function of these buttons like the function of the SECTION tab

3.4.2 PANEL Defaults

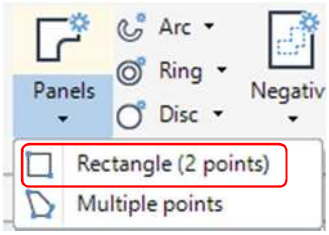


Use to sets the default specification for the creation of panels. The default specification field values can also be applied to existing panels.

	**Panel classification:** Input a value or select the default panel classification from the drop-down list to be used for the creation of panels.
	**Default Material:** Input a value or select the default material from the drop-down list to be used for the creation of panels.
	**Default Justification:** To choose Top (TOP), Centre (CENT) or Bottom (BOTT) of the panel
	**Specification:** Choose standard of panel
	**Panel Thickness:** Input a value or select the default thickness from the drop-down list to be used for the creation of panels.
	**Drawing Edge Defaults:** Allows you to define the default drawing edge defaults for panel elements.

3.4.3 Create group from the PANEL tab

3.4.3.1 Create Rectangle Panel

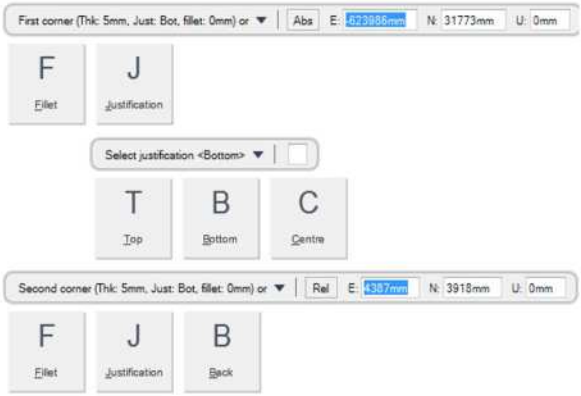


Creates a rectangle panel with two defined points.

First: Choose button **Rectangle**

Second: Input values in the dimension fields and press Enter or click to identify the first point position.

Press the down cursor key displays the options below



Fillet: Input a dimensional value in the field and press Enter to set the fillet of the panel.

Justification: Set up justification for panel. Press the down cursor key choose select justification

Top: To create a panel with top justification.

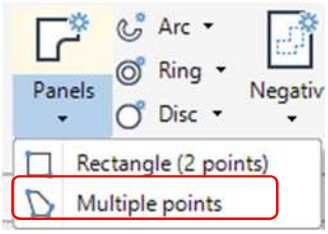
Bottom: To create a panel with bottom justification.

Centre: To create a panel with center justification.

Back: To undo the last action.

Third: Input values in the dimension fields and press Enter or click to identify the second corner position

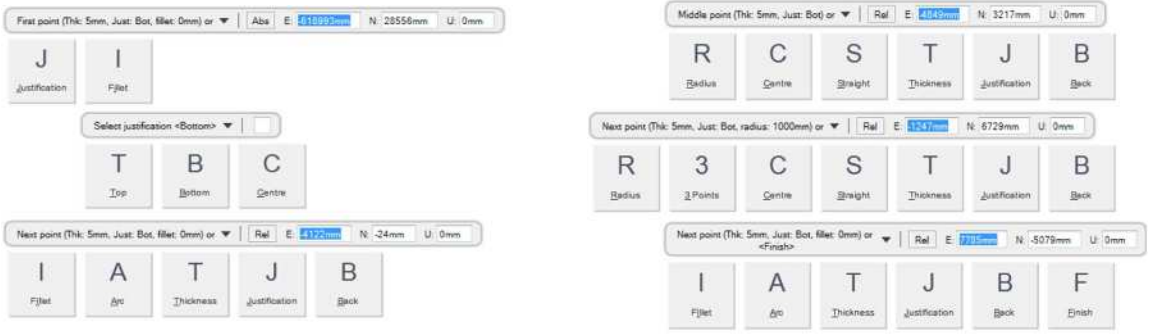
3.4.3.2 Create Multiple Point Panel



Creates a multiple point panel.

First: Choose button **Multiple Point**

Second: Input values in the dimension fields and press Enter or click to identify the first point position.Press the down cursor key displays the options below



Fillet: Input a dimensional value in the field and press Enter to set the fillet of the panel.

Justification: Set up justification for panel. Press the down cursor key choose select justification

Top: To create a panel with top justification.

Bottom: To create a panel with bottom justification.

Centre: To create a panel with center justification.

Arc: Select Arc to create a 3 points, Radius or Centre curved panel edge.

Thickness: Select Thickness to define the thickness of the panel.

Radius: Select Radius to create a panel edge with a specified radius.

Straight: Select Straight to create a straight panel edge.

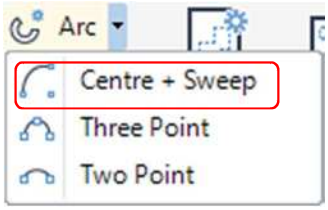
3 points: Select 3 points to create a curved panel edge with 3 points.

Finish: Select Finish to end the command.

Back: To undo the last action.

Third: Select Finish to end the command.

3.4.3.3 Create Arc Panel with Centre Point and Sweep Angle

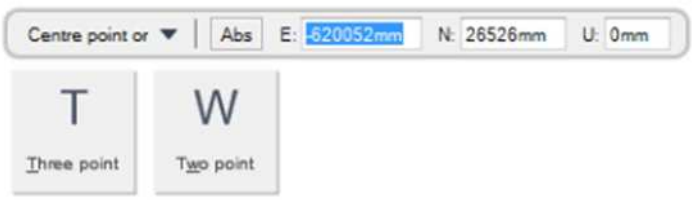


Creates an arc panel with a specified center point and sweep angle.

OPTION 1:

First: Choose button **Centre + Sweep**

Second: Press the down cursor key displays the options below



Three point: Select Three point to create an arc with three specified

Two point: Select Two point to create an arc with two specified points.

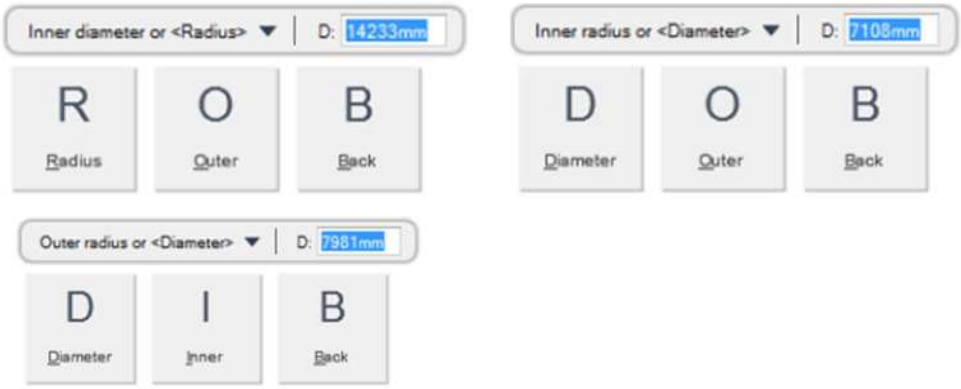
Third: Choose specify panel width and finish

OPTION 2:

First: Choose button **Centre + Sweep**

Second: Input values in the dimension fields and press Enter or click to identify the center point position.

Press the down cursor key displays the options below



Radius: Select Radius to create an arc panel with a fixed inner or outer radius.

Diameter: Select Diameter to create an arc panel with a fixed inner or outer diameter.

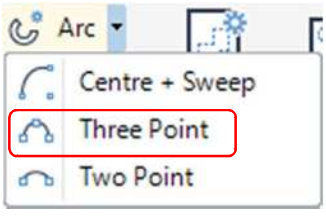
Outer: Select Outer to create an arc panel with a defined outer diameter or radius.

Inner: Select Inner to create an arc panel with a defined inner diameter or radius.

Back: Select Back to undo the last action.

Third: Input start angle or <0deg> 250 and then input sweep angle or <0deg>120 and finish

3.4.3.4 Create Arc Panel with Three Points



Creates an arc panel with three specified points.

OPTION 1:

First: Choose button **Three Point**

Second: Press the down cursor key displays the options below to choose the option to below

Centre: To create a panel with center justification.

Two point: Select Two point to create an arc with two specified points.

Next step: To the same to creat Centre + Sweep

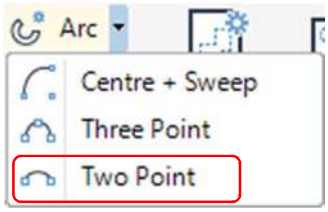
OPTION 2:

First: Choose button **Centre + Sweep**

Second: Choose the first point, the second point and then the last point.

Third: Choose specify panel width and finish

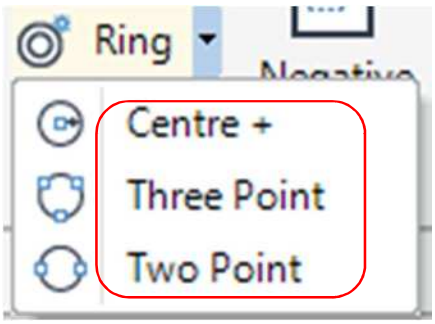
3.4.3.5 Create Arc Panel with Two Points



Creates an arc panel with two specified points.

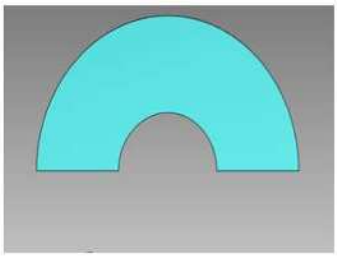
Create is the same is **Three Point** but in the step second of **OPTION 2:** Just choose two point

3.4.3.6 Create Ring Panel with Specified Centre Point, Three Point and Two Point.

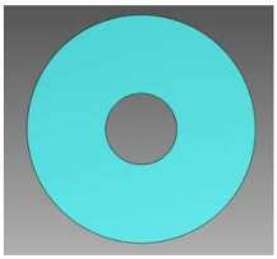


Creates a ring panel with a specified centre point

Steps to create functions of the Ring panel are the same as the function to create Arc panel Centre Point and Sweep Angle, Three Points Two Points. But is different to complete product like to picture below :

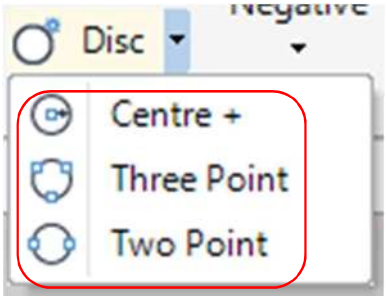


Product of Arc Panel



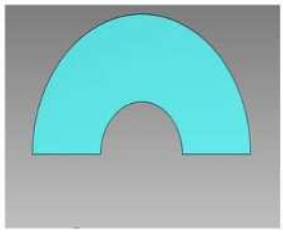
Product of Ring Panel

3.4.3.7 Create Disc Panel with Specified Centre Point, Three Point and Two Point.

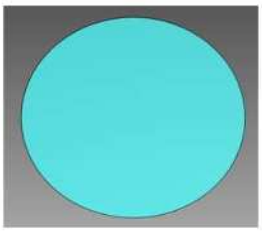


Creates a disc panel with a specified center point.

Steps to create functions of the Disc panel are the same as the function to create Arc panel Centre Point and Sweep Angle, Three Points Two Points. But is different to complete product like to picture below :

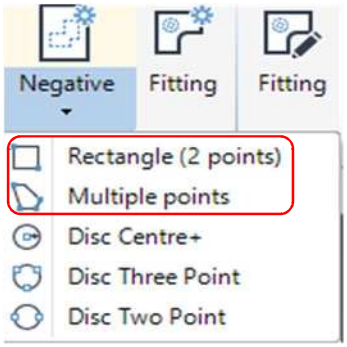


Product of Arc Panel



Product of Disc Panel

3.4.3.8 Create Negative Panel with Rectangle and Multiple point.

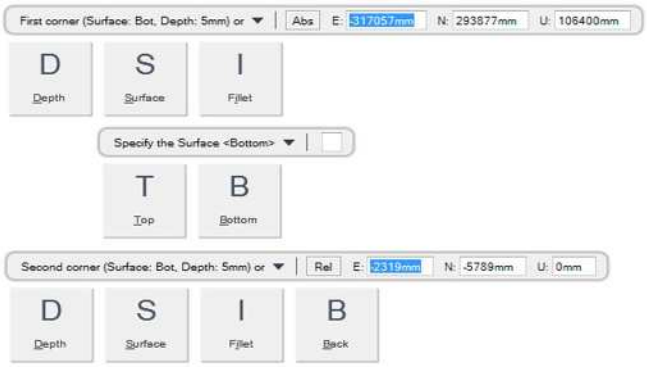


Creates a hole rectangle on the panel with two defined points.

First: Choose button **Rectangle** or **Multiple point**

Second: Input values in the dimension fields and press Enter or click to identify the first point position.

Press the down cursor key displays the options below



NOTE: With button Rectangle you just choose two point to make a hole on the panel and with button Multiple points you can choose multiple points to make a hole on the panel.

Depth: Select Depth to define the depth of the negative extrusion.

Surface: Select Surface to define the surface position of the negative extrusion.

Fillet: Select Fillet to create a negative extrusion with a specified fillet.

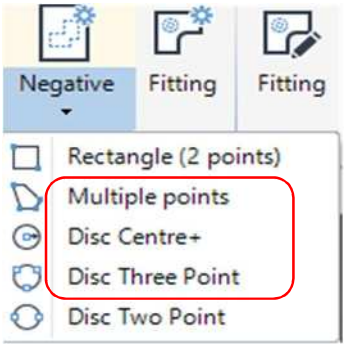
Top: Select Top to create a negative extrusion aligned to the top of the

Bottom: Select Bottom to create a negative extrusion aligned to the bottom of the associated panel.

Back: To undo the last action.

Third: Input values in the dimension fields and press Enter or click to identify the second point position and finish.

3.4.3.9 Create Disc Centre+, Disc Three Point and Disc Two Point



Creates a hole Disc Centre+, Disc Three Point and Disc Two Point on the panel.

Steps to create functions of the Disc Centre+, Disc Three Point and Disc Two Point are the same as the function to create **Rectangle** But is some diffirent like below:

With **Rectangle:** Select two point

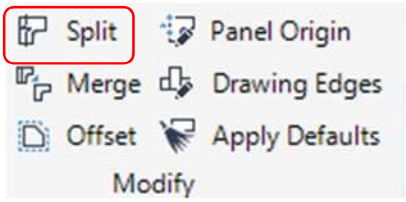
With **Disc Centre+**: Select center point and input radius to make hole finish

Disc **Three Point**: Select three point to make hole and finish

Disc **Two Point**: Select two point and to make hole and finish

3.4.4 Modify group from the PANEL tab

3.4.4.1 Modify Split Panel



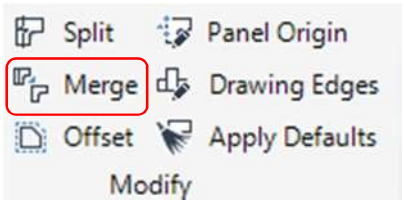
Splits a panel to be two or multiple panels

First: Choose button **Split** and then choose a panel

Second: Press the down cursor key and click Next. Select the element to split or press the down cursor key and click Plane. Input the xyz coordinates to first point or choose pick point and then input the xyz coordinates to the second point or choose to pick to make the plane to split.

Third: Press the down cursor key and click Finish or you can choose third point to finish.

3.4.4.2 Modify Merge Panel

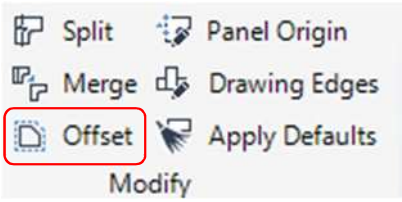


Merges two panels to be a panel

Choose button **Merge,** choose first panel and then choose two panels or multiple. Press the down cursor key and click Finish

NOTE: Default justification all panels must be the same and lying on a plane
The properties of the merged panels are like from the first picked panel.
Panels to merge must be touch together and not have distance.

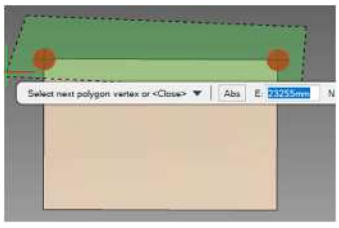
3.4.4.3 Modify Offsset Panel



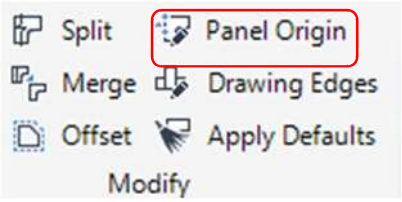
First: Choose button **Offset** and then choose a panel

Second: Press the down cursor key and click All to offset all pave of panel and input value in the field

Press the down cursor key and click Select vertices and then draw a polyline to choose paves of the panel you want to offset after that press the down curso200r key and click close and input value in the field



3.4.4.4 Modify Panel Origin

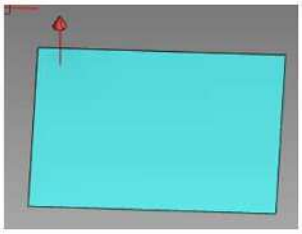


When creating a panel the Origin is placed at the first vertex by default and you modify split, merge or offset … the origin of the panel not exactly like the first time. This button to modify the origin again.

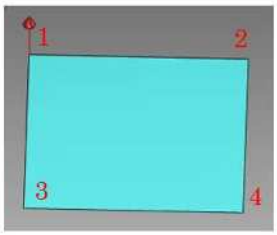
Choose button **Panel Origin** and then choose a panel to modify.

NOTE: If you want to the origin of the panel near position you click mouse nearly that pave and then the origin will be at that pave.

Example: Picture 1 not yet modify origin. Picture 2 modified if you click mouse near position 1, 2, 4 and 4 the origin will be there

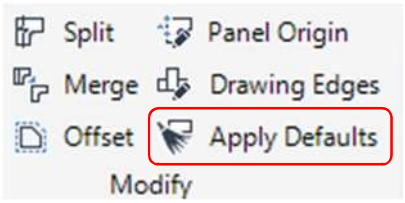


Picture 1



Picture 2

3.4.4.5 Modify Apply Defaults

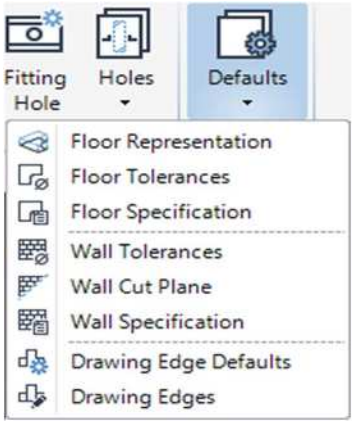


Apply all seting with the panel. It use to modify specification, default jusfitication or thickness … of the panel.

3.5 WALL AND FLOORS tab

3.5.1 Create table

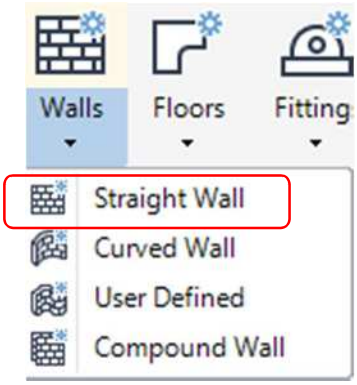
3.5.1.1 Defaults Settings



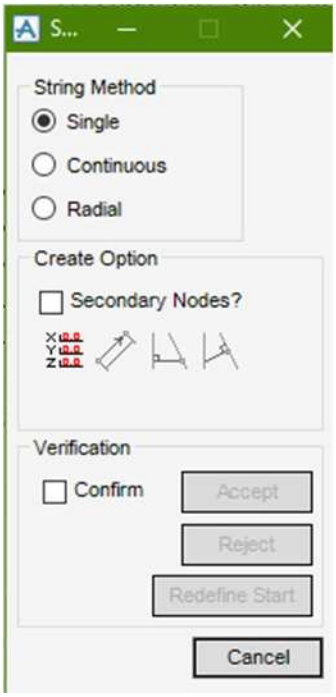
Sets the default specification for the creation of walls and floors. Before walls and floors can be designed as part of the overall 3D model, various options must be defined which determine.

Floor Representation	Modifies the level and obstruction which affect the way elements are displayed in the 3D view
Floor Tolerances	Modifies the tolerances (clearances) of floor elements to suit project/model requirements.
Floor Specification	Sets the floor/screed elements available for selection from the structural catalog as determined by the default specification. The default specification defines properties such as material and cross-sectional profiles.
Wall Tolerances	When a joint which has one or more attached sections is repositioned, the effect on those sections depends upon whether or not the joint has been defined as dominant or subordinate, as defined by the setting of the joint's Joint Freedom (JFRE) attribute.
Wall Cut Plane	Specifies the start or end of the wall's cut plane with the use of selection functionality
Wall Specification	Sets the wall elements available for selection from the structural catalogue as determined by the default specification. The default specification defines properties such as material and cross-sectional profiles

3.5.1.2 Create Straight Wall



First: Choose button **Straight wall** to show the table below. The **Positioning Control window** also displays, the positioning functionality available for selection is common throughout all design disciplines.



Second: You have to three options available you choose one of three option to creat straight wall

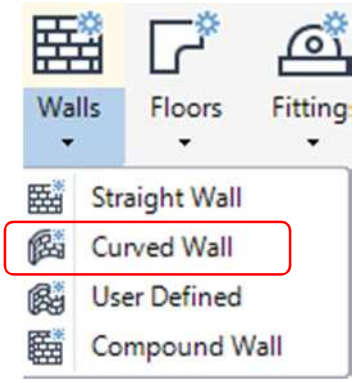
Single - Start and end points are individually defined for each section

Continuous - Start point of subsequent walls is the end point of previous

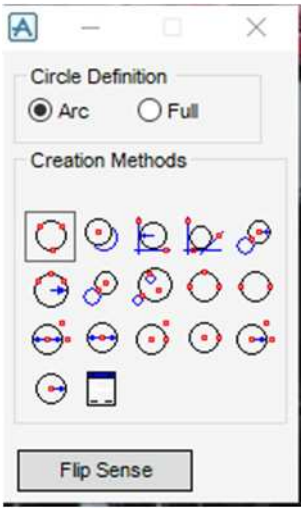
Radial - Start point for subsequent walls is the same as the first

NOTE: The default height of wall create is 3000mm and to change the height to change desparam

3.5.1.3 Create Curved Wall



First: Choose button **Curved wall** to show the table below. The **Positioning Control window** also displays, the positioning functionality available for selection is common throughout all design disciplines.

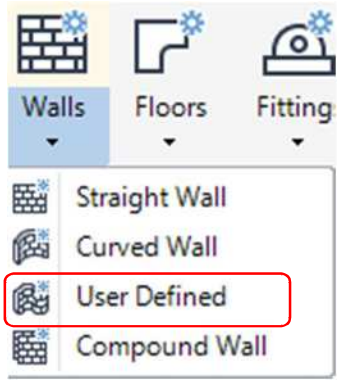


Second: Choose the option in Circle Definition table to make new the Curved Wall

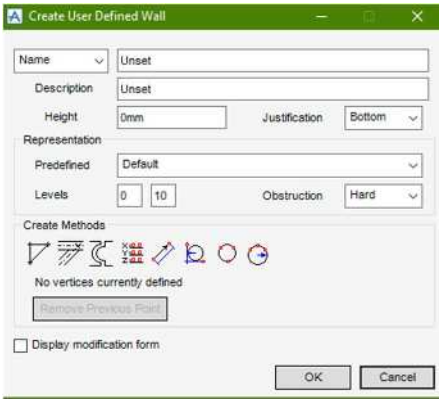
	**Through 3 points**	Click to create a curved wall which passes through three defined points. You are prompted to define three points in the 3D view.
	**Derived from a picked item**	Click to create an offset ring wall on a structure that already exists. You are prompted to pick an item in the 3D view.
	**Fillet**	Click to create a curved wall with a specified radius between two picked tangent lines.
	**Tangential to 3 lines**	Click to creates a curved wall which forms the tangents to three straight lines.
	**Fixed radius, tangential to a circle**	Click to create a curved wall with a specified radius which touches another ring wall at one point.
	**Fixed radius, passing through 2 points**	Click to create a curved wall with a specified radius which passes through two picked points and curves towards a picked direction

	**Tangential to a circle**	Click to create a curved wall which intersects a ring wall that already exists
	**Tangential to 2 circles**	Click to create a curved wall which touches two other ring walls tangentially.
	**Derived diameter**	Click to create a curved wall from three defined points
	**Derived diameter on working plane**	Click to create a curved wall on the working plane. The wall is created irrespective of the distance above the working plane and the wall parameters are defined
	**Fixed diameter**	Click to create a curved wall with a fixed diameter around a single point.
	**Fixed diameter on working plane**	Click to create a curved wall with a fixed diameter around a single point. The wall is created on the working plane irrespective of the distance the single point is above the working plane.
	**Derived radius**	Click to create a curved wall to a predetermined radius.
	**Derived radius on the working plane**	Click to create a curved wall to a predetermined radius. The wall is created on the working plane irrespective of the distance the predetermined radius points are above the working plane.
	**Fixed radius**	Click to create a ring wall with a fixed radius.
	**Fixed radius on the working plane**	Click to create a curved wall with a fixed radius. The wall is created on the working plane irrespective of the distance the fixed radius points are above the working plane.
	Flip Sense	To flip the ring wall 90 degrees

3.5.1.4 Create User Defined Wall



First: Choose button **User Defined** to show the table below



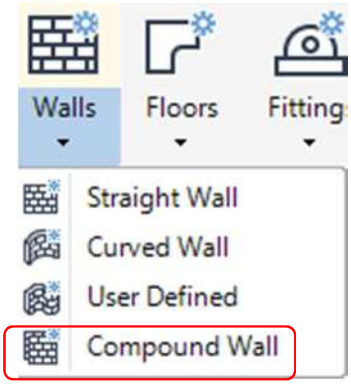
You must set height, justification, and representation before go to step second

Second: Choose the option in Create Methods table to make new the User Defined Wall

	**Derived points from graphic picks**	Allows you to pick a point graphically with the **Positioning Control** window.
	**Derive points from centres & plines**	Allows you to construct a point from the intersection of two Plines with a floor that already exists.
	**Trace boundary of another extrusion/panel**	Allows you to create vertices which follow the shape of a floor that already exists.
	**Explicitly defined position**	Allows you to specify a position by entering explicit co-ordinates.
	**Point offset from previous**	Allows you to specify a distance and direction which define a point relative to the position of the vertex that precedes it.
	**Define arc fillet**	Allows you to construct a fillet arc with a specified radius between two picked tangent lines.

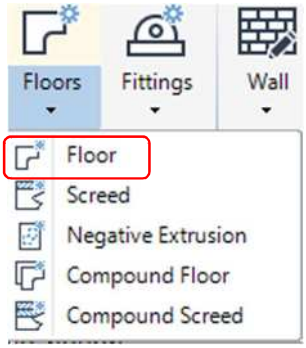
	**Derived arc passing through three points**	Allows you to construct a fillet arc that passes through three picked points.
	**Fixed radius arc, passing through 2 points**	Allows you to construct a fillet arc with a specified radius, that passes through two picked points, which curves towards a picked direction.

3.5.1.5 Create Compound Wall

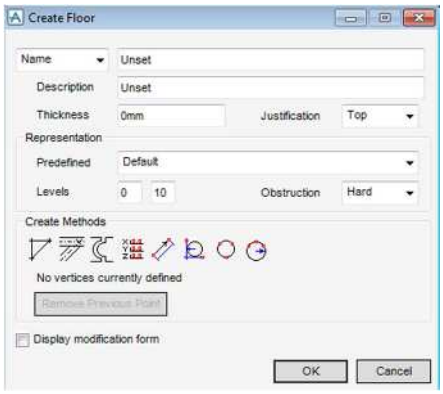


Choose button **Compound Wall** to show the table below and then input name. Compound Wall used to the same to sub-frame in the design hierarchy.

3.5.1.6 Create Floor



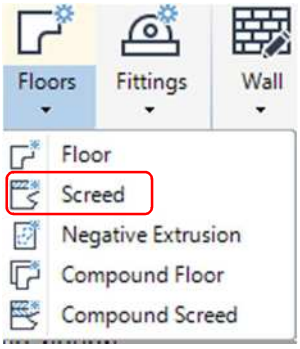
First: Choose button **Floor** to show the table below



Second: Input thickness, justification, and representation

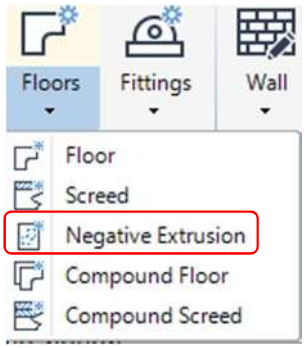
To make Creat Methods is the same in User Defined Wall

3.5.1.7 Create Screed



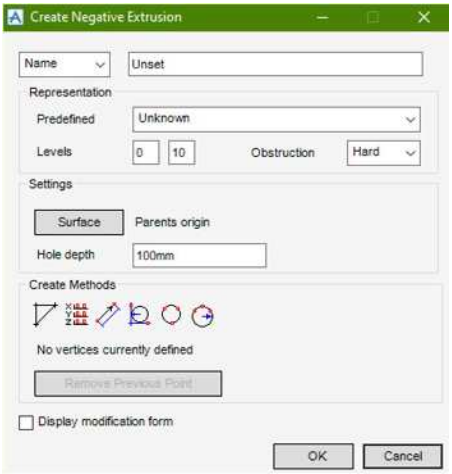
The functionality available from the Create Screed window is identical to the functionality available from the Create Floor

3.5.1.8 Create Negative Extrusion



Creates a Negative Extrusion (NXTR) by positioning PAVE elements to form a Loop (LOOP) and then create a hole in the floor.

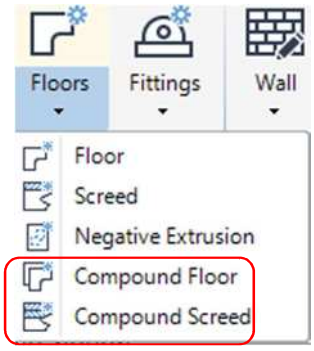
First: Choose the Floor and click the button **Negative Extrusion** to show the table below



Second: Click Surface. Then pick the upper face of the floor/screed. Input a hole depth

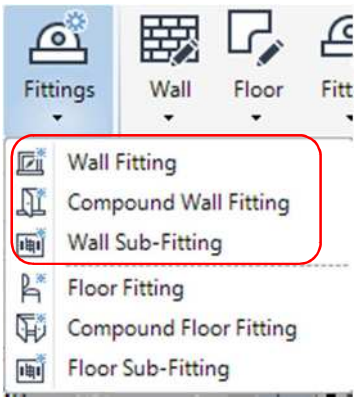
And then Create Methods is the same in create Floor

3.5.1.9 Create Compound Floor and Compound Screed



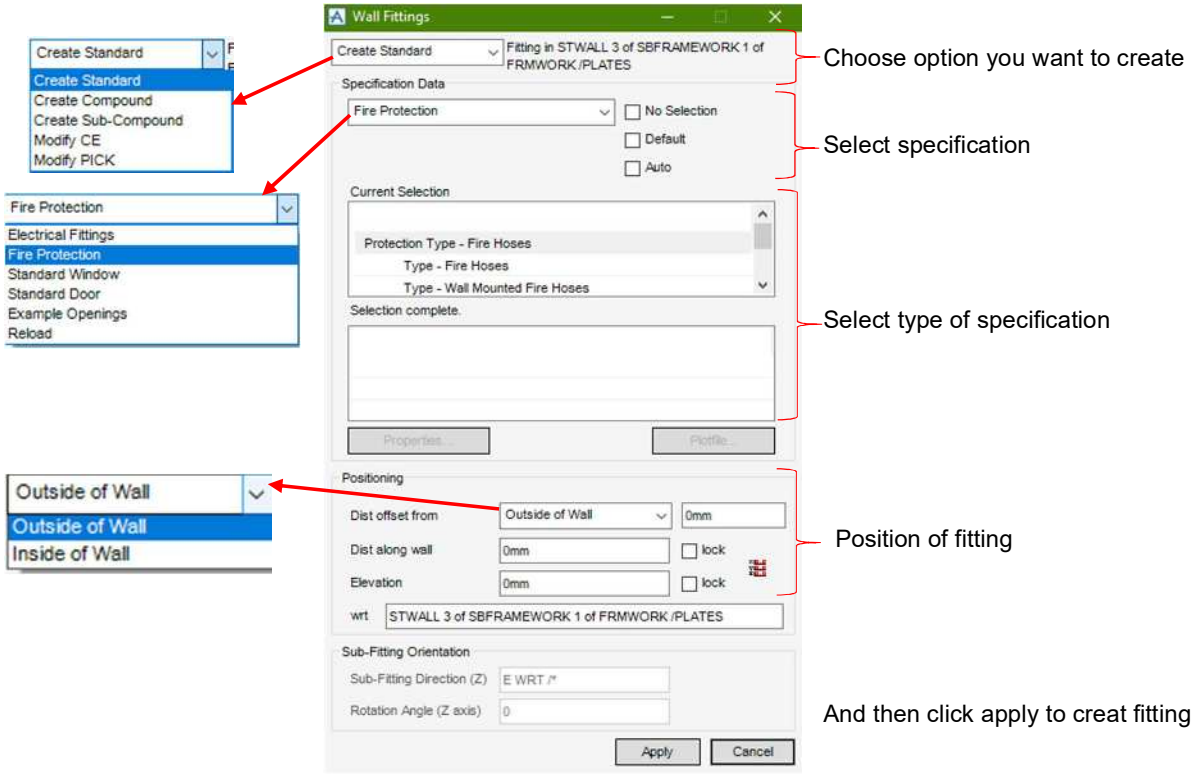
Choose button **Compound Floor or Compound Screed** to show the table create name and then input new name. It used to the same to sub-frame in the design hierarchy.

3.5.1.10 Create Fittings



To create new wall fitting, compound wall fitting and wall sub-fitting on the wall.

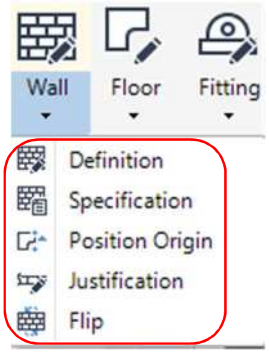
Click one of three button above to show the table below



Step to create new floor fitting, compound floor fitting and floor sub-fitting on the floor is the same to create fitting on the wall above

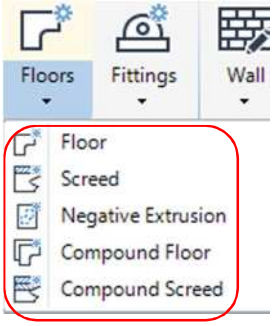
3.5.2 Modify table

3.5.2.1 Modify Wall



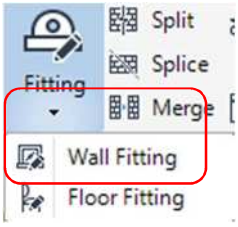
Definition	Modifies the position, length and direction of a straight wall.
Specification	Modifies the wall specification using available elements from the structural catalogue
Position Origin	Modifies the vertex position of the wall origin.
Justification	Modifies the wall pline justification
Flip	Flips the wall element 180 degrees.

3.5.2.2 Modify Floor



Definition	To modify Vertex Editor allows you to move, add and delete vertices, edges and arcs of floor, screed
Justification	Modify Justification bottom, center or top of the floor, screed
Thickness	Modifies the thickness attribute of a floor/screed element.
Negative Extrusion	To modify Vertex Editor allows you to move, add and delete vertices, edges and arcs of Floor/Screed Negative Extrusion
Position Origin	Modifies the vertex position of the floor/screed origin.
Specification	Modifies the floor/screed specification using available elements from the structural catalogue

3.5.2.3 Modify Fitting



Wall Fitting: Modifies the catalogue specification associated with each wall fitting.

Floor Fitting: Modifies the catalogue specification associated with each floor or screed fitting.

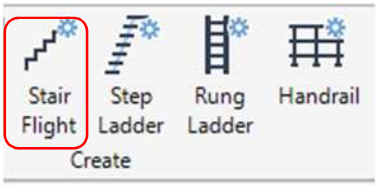
3.5.2.4 Some Tools Modify

Split		Splits a straight wall into shorter lengths where it intersects with another element.
Splice		Divides a wall into smaller lengths using a splice. You must specify the plane to create the splice.
Merge		Merges individual colinear walls to form a single wall.
Mitre Ends		Mitres a straight wall end to another straight wall end. You must pick two walls to be mitred, once both walls are picked, the walls are mitred automa
Wall Ends	Extend Through	Sets a plane, position and orientation to extend either end of the wall to. The wall only extends along its Z axis, that is its extrusion direction, as the plane is infinite in its planar dimensions.
	Extend By	Lengthens or shortens a wall by moving the start or end position by a specified amount.
	Trim End	Trims the length of an attached wall to intersect with an owning wall if a wall position has been modified.
	Trim Connected	Trims the length of all attached wall ends to intersect with a selected owning wall if a wall position has been modified.
Split Floor		Splits a floor or a screed into multiple floors.
Screed Slope		Applies a single slope to a screed element with the addition of a negative volume to the screed definition. Only one slope can be applied to any screed element, to apply multiple slopes to an area, you must first split the overall screed into separate elements.

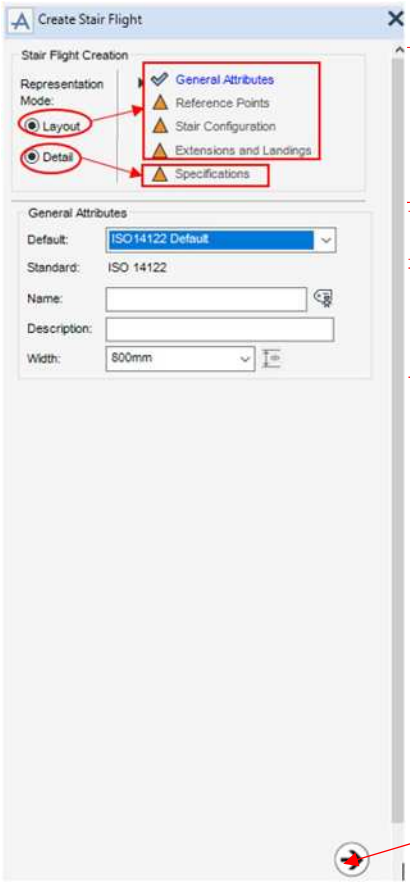
3.6 STAIRS LADDER AND HANDRAILS tab

3.6.1 Create table

3.6.1.1 Stair Flight



Choose button **Stair Flight** to show the table Create Stair Flight below



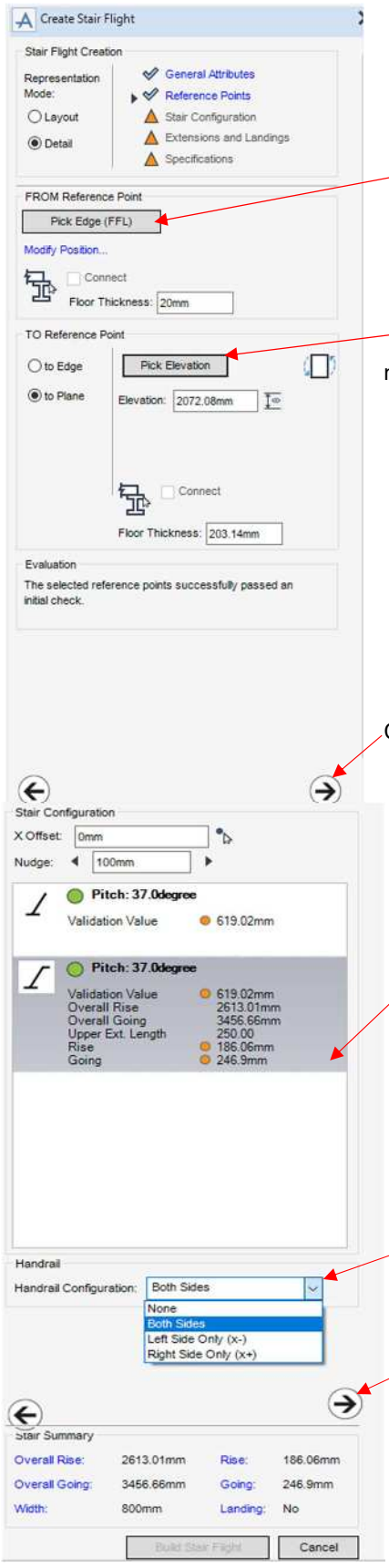
Step 1:

Choose the layout you have to 4 step from general attributes to extension and landings and if you choose the detail you have more step specifications. And if you finish the color of the text is blue and have a sticker ð

Choose default of stair flight

Input name, description and width of stair flight

Click here go to Step 2



Step 2

Click here and then choose the top floor to make the top of the stair

Click choose Plane and click here after that choose the plan or floor to make the bottom of the stair

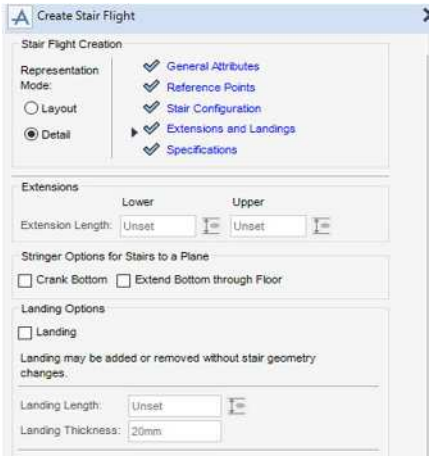
Click here go to Step 3

Step 3

Click here and then choose type of the stair flight

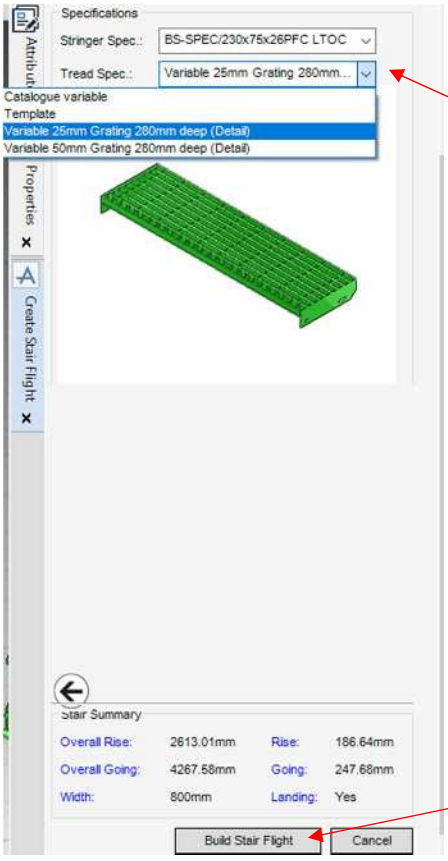
Click here to choose the option of the handrail

Click here go to Step 4



Step 4

Chose some option you need to create the stair flight and click



Step 5

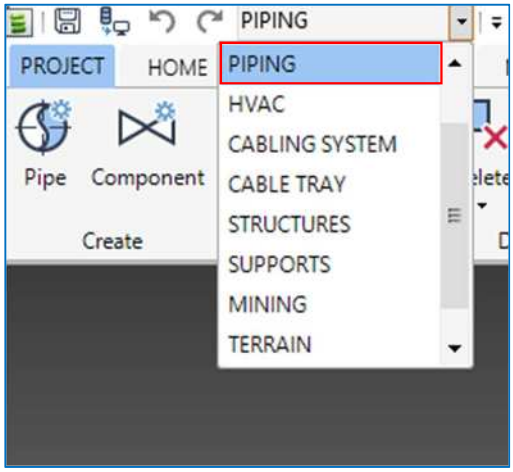
Click here to choose specifications of a plate of stair

Click here to create the Stair Flight

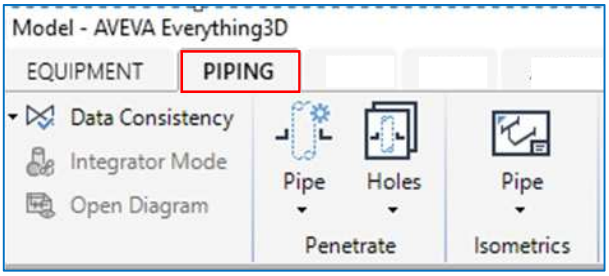
4. Piping

4.1 Getting to start

Launch AVEVA E3D then from the application options list select PIPING.



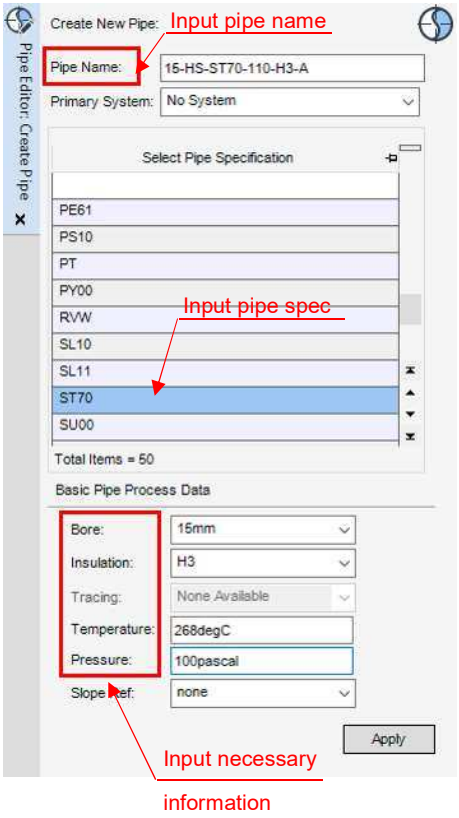
After selecting PIPING module, the PIPING tab will available as picture below:



4.2. Work with Piping Group

4.2.1 Create Group

4.2.1.1 Creating Pipe



In the **Create** group click the **Create Pipe** button to display the **Pipe Editor: Create Pipe** form.

The **Pipe Editor: Create Pipe** form is now displayed. The upper section of the form allows the **Pipe Name** to be entered and the **Primary System** to be selected.

The middle section of the form allows the selection of the specification for the pipe.

The lower section of the form is the **Basic Pipe Process Data**

- **Bore**
- **Insulation Spec**
- **Temperature Spec**
- **Pressure**

Not all of these fields are mandatory.

Clicking the **Apply** button will create the pipe which in turn changes the form to the **Pipe Editor: Modify Pipe** form (see 1.2.2).

4.2.1.2 Creating Branch

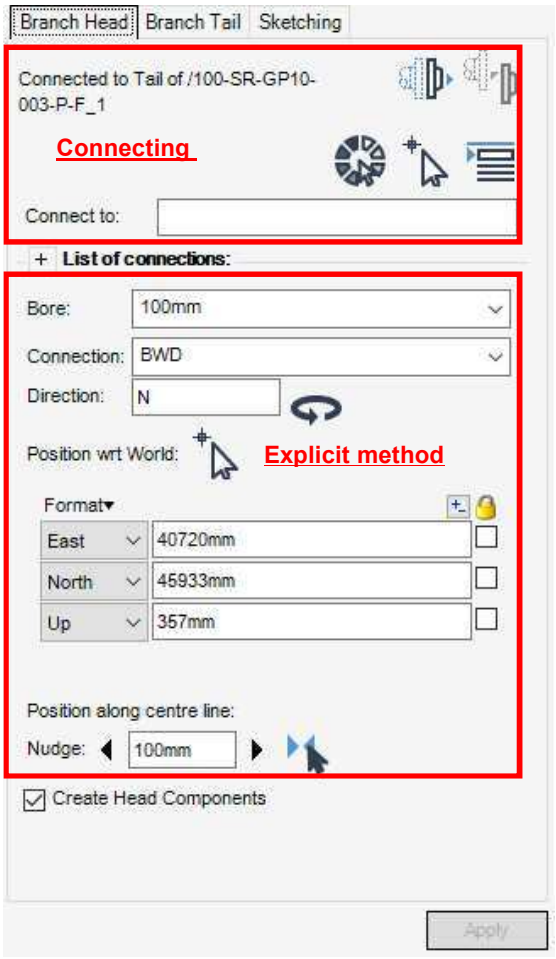
The **lower section** of the **Pipe Editor: Modify** form contain the branch manipulating process.

Clicking **Create a new branch** button to create new branch belonging to the current pipe.

Branch Head/Tail Positioned.

There are two methods to set the position and connection for branch:

- Explicitly – by entering the data manually
- Connecting – by connecting to an existing pipe or equipment nozzle the values are set automatically



For Connecting method

Disconnect – disconnects the head/tail from the current connection.

Reconnect – reconnects the head/tail to a connection that has changed position in the model.

Pick item to connect – connect the head/tail to picked nozzle (if the nozzle has been connected before, the process will be canceled).

Connect to last/first member – connect head/tail to first/last member.

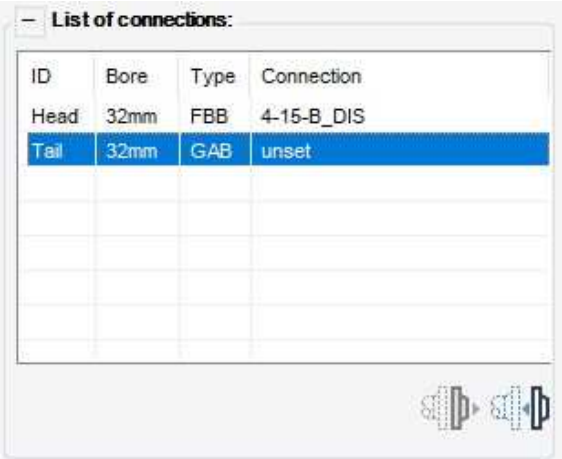
For Explicit method

User needs to input **Bore, Connection, Direction, Position** of head/tail manually.

Pick 3D point – pick a point in 3D View to define position of head/tail.

Positioning along center line – change explicit point by moving head/tail along PH/PT.

Create Head/Tail Components – create gasket and flange automatically when connect head/tail to flanged items.



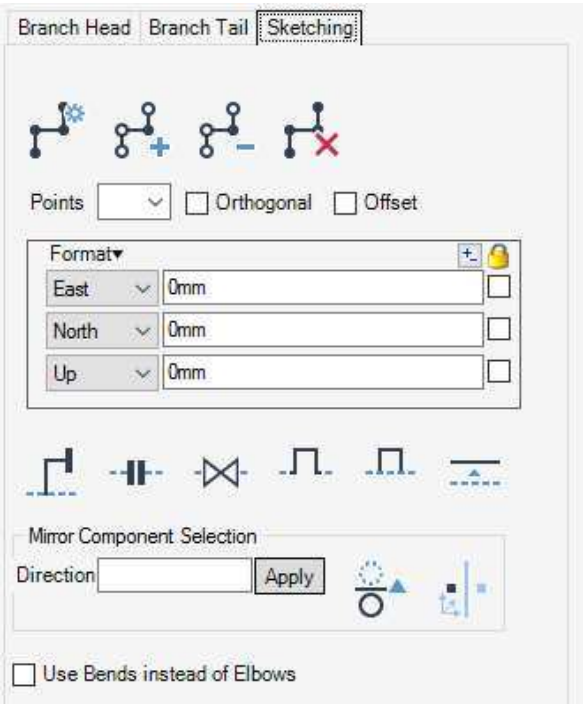
List of connections – show head/tail information of connected item.

Disconnect connected item – disconnect the selected item.

Connect to selected item – connect head/tail to selected item when the item is unconnected.

Pipe Sketching.

Use the pipe sketching functionality to access design aids and tools to create a branch, insert components, copy and mirror piping components.



Aid Points

Use the aid point functionality to create or add aid points to a branch.

Create Point – add aid points in 3D view.

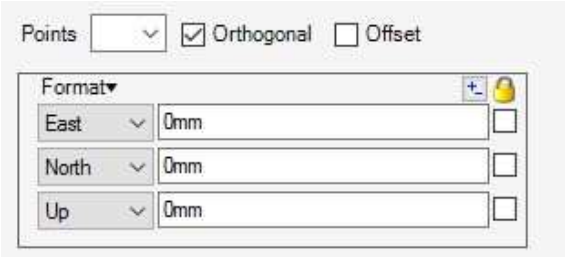
Add Point – add an additional aid point in 3D view.

Delete – delete the last aid point from 3D view.

Delete All Aid Points – delete all the aid points.

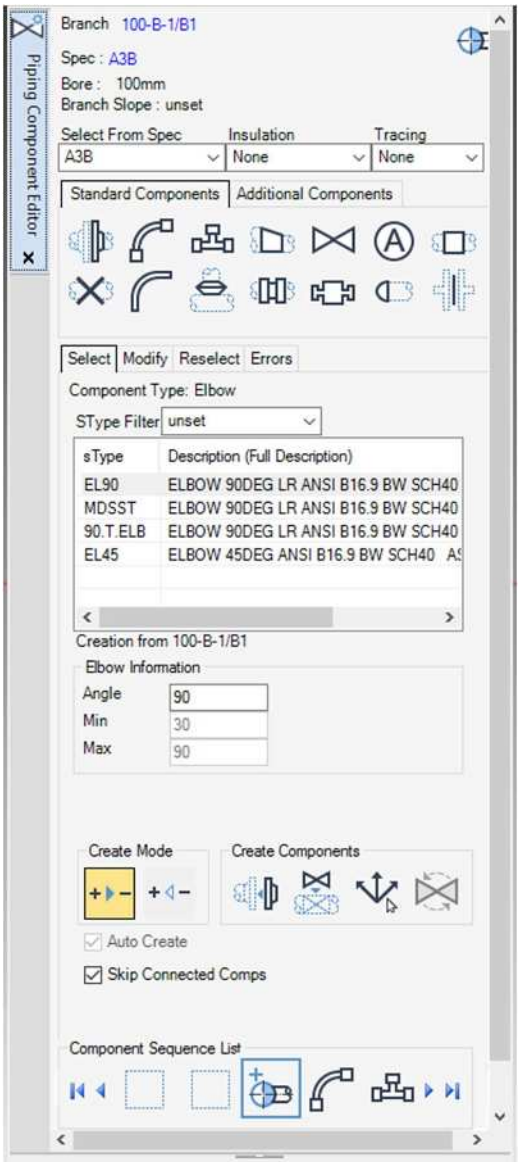
Position aid points

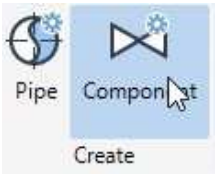
Select an aid point from the **Points** drop down list, to view the frame, offset and world position of an aid point. Use the **orthogonal** functionality to view the line between the aid points in the 3D view as orthogonal. Use the offset functionality to offset an aid point in the 3D graphical view.



	**Create Branch**	Use aid points to create a new branch in 3D view. The new branch attributes are based on the currently selected branch in the hierarchy. Depending upon configuration of the aid points, a **CHOOSE** window is displayed listing a selection of piping components. If an intersection between the aid points and the currently selected branch exists, the new branch is added to the current branch. If required the new branch can be created at an offset distance from the currently selected branch.
	**Insert Flange Set**	Use to create a flange set at the intersection of an aid point line and an existing branch. Once the selection of the intersecting pipe in the 3D view has been made, a series of **CHOOSE** window are displayed listing the available flanges and gaskets. The flange set is then added to the branch at the intersection with the aid point.
	**Insert Valve Set**	Use to create a valve set at the intersection of an aid point and an existing branch. Once the selection of the intersecting pipe in the 3D view has been made, a series of **CHOOSE** window are displayed listing the available flanges, gasket and valves. The valve set is then added to the branch at the intersection with the aid point.
	**Insert Expansion Loop**	Use to create an expansion loop at the intersection of an aid point line and an existing branch. Once the selection of the intersecting pipe in the 3D view has been made, a **CHOOSE** window is displayed listing the available elbows. The expansion loop is then added to the branch at the intersection with the aid point.
	**Insert Bypass**	Use to create a bypass at the intersection of an aid point line and an existing branch. Once the selection of the intersecting pipe in the 3D view has been made, a series of **CHOOE** window are displayed listing the available tees and elbows. The expansion loop is then added to the branch at the intersection with the aid point.
	**Copy Selection**	Use to copy one or more branch elements and position them at a new location in the currently selected branch (make sure that all the components are connected and there are no unwanted components). **Note:** The source branch component is usually the origin component from the selection.
	Direction	Use to enter a direction for the mirror into the **Direction** field. Once an element has been picked in the 3D view, a mirror plane is displayed in the 3D view.
	**Pick Mirror Direction**	Use the Pick mirror Direction functionality to Pick a direction in the 3D view. After an element has been picked in the 3D view, a mirror plane is displayed in the 3D view.
	**Pick Mirror Position**	Use to pick a mirror position in the 3D view. Once selected the **Positioning control** toolbar is displayed, Use its functionality to position the mirror plane.
	Use Bends instead of Elbows	Replaces bends instead of elbows in the 3D view.

4.2.1.3 Creating Piping Component





In the **Create** group click the **Create Component** button to display the **Piping Component Editor** form.

The **Piping Component Editor** form now appears.

This form is used extensively during pipe creation. As implied by the name of the form it is not only restricted to the creation of the components, but also for the modification and reselection as well.

Branch Specification

The **Spec:, Bore:** and **Branch Slope**: of the currently selected pipe displays at the top of the window.

Select a different specification for the component, the branch specification is initially selected but an alternative specification can be selected.

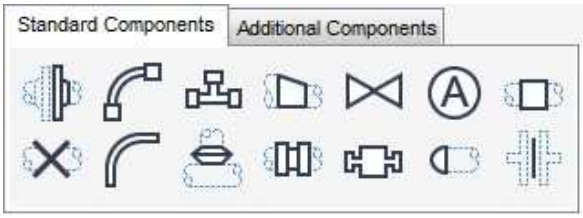
Select a specification form the **Select From Spec** drop-down list.

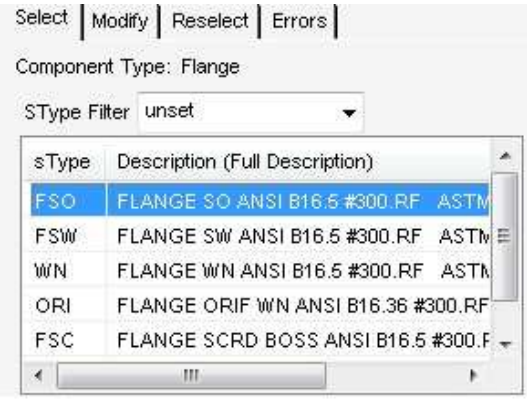
If a specification is selected that does not match the pipe specification, **The Piping Component Editor** window displays a **Select Out of Spec** message.

Repeat the same step for **Insulation** and **Tracing**.

Standard Components.

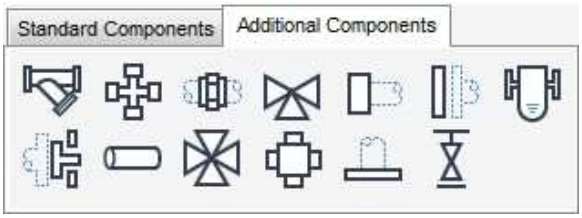
The **Standard components** tab (displayed as default) provides quick access to the more commonly used components.





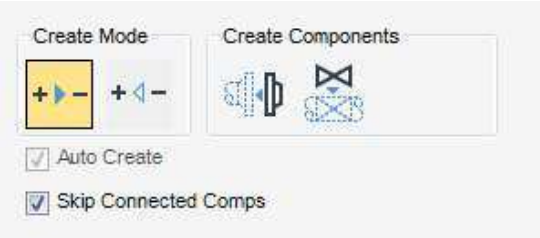
The **Select** tab displays all of the components available for selection from the catalogue under the currently selected component and piping specification.

Additional Components.



The **Additional Components** tab provides quick access to further components such as: filter, cross, union, three way valve, closure, Blind flange, trap, lap joint, Ftube, four way valve, shu, plate…

Create Mode.



Use the **Create Mode** part of the **Piping Component Editor** window to set the direction of flow, the software creates the component in the head to tail direction by default (corresponding to **forward** and **backward mode**)

	**Create Components in the flow direction**	Creates the component in the head to tail direction.
	**Create components against the flow direction**	Creates the component in the tail to head direction.

Note: if the **Auto Create** check box is selected, the software automatically determines if there is a requirement for an associated component (on a vale, the associated component would be a flange and gasket) and if so, what is the appropriate.

If The **Out of Spec** check box is selected, an alternative specification component can be used.

If the **Skip Connected Comps** check box is selected, the pipework application automatically determines which components are connected, then positions the creation point approximately.

Create Component.

Use the **Create Components** part of the **Piping Component Editor** window to connect or place the component.

	**Create component connected to previous**	Connects the component to the previous one or branch head/tail.
	**Create Component in-line at picked position**	Positions the component at the point selected in the 3D view
	**Create component at 3D picked position**	Insert new component after the current element in the branch. Its position is determined by selection a position in the 3D view.

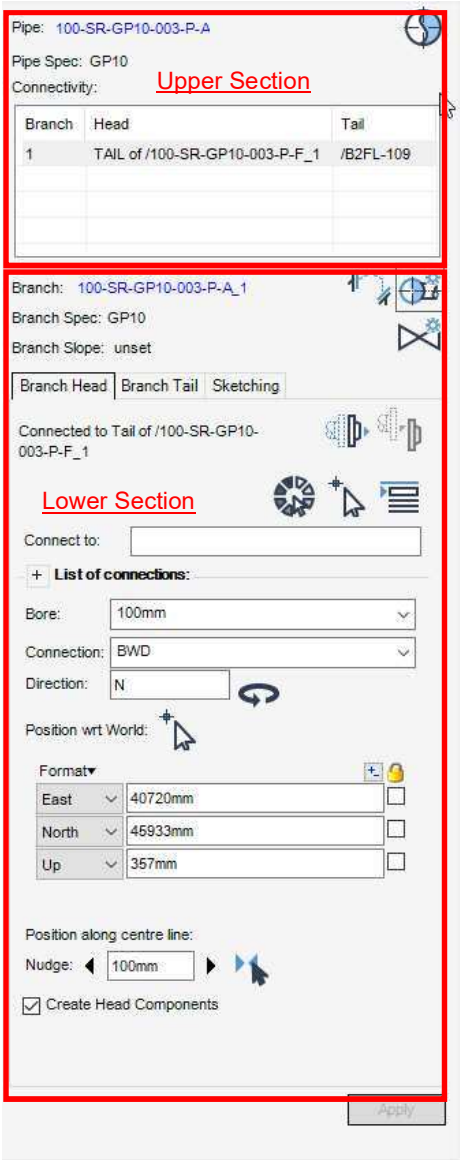
Component Sequence List.

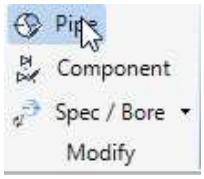
Use the **Component Sequence List** part of the **Piping component Editor,** to scroll through and select individual components within the currently selected branch.

	Moves the start of the Component Sequence List.
	Moves one component to the left of the Component Sequence List.
	Moves one component to the right of the Component Sequence List.
	Moves to the end of the Component Sequence List.

4.2.2. Modify Group

4.2.2.1 Modifying Pipe





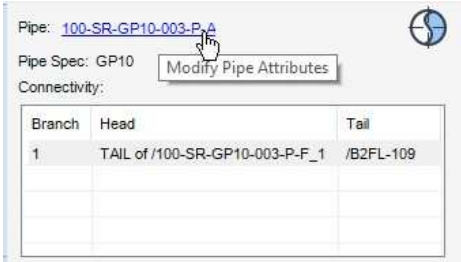
In the **Modify** group click the **Modify Pipe** button to display the **Pipe Editor: Modify Pipe** form. Or clicking the **Apply** button on the **Pipe Editor: Create Pipe** form, form the **Pipe Editor: Modify Pipe** is automatically displayed so that the branch head and tail can be specified.

The **upper section** of the displays the pipe name and specification. There is also a list of existing branches containing the reference names for head and tail connections.

The **lower section** of the form shows the Branch Head and Branch Tail tabs. These allow the positioning, connection and setting of various other attributes for the head and tail of the selected branch.

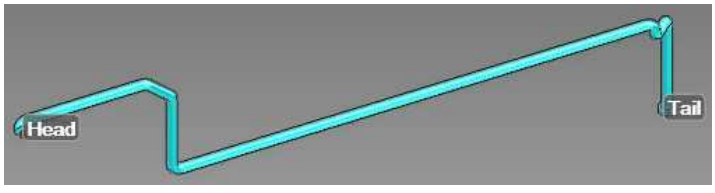
It can be considered that there are two methods of doing this:

- Explicit – by entering the data manually.
- Connecting – by connecting to an existing pipe or equipment nozzle the values are set automatically.

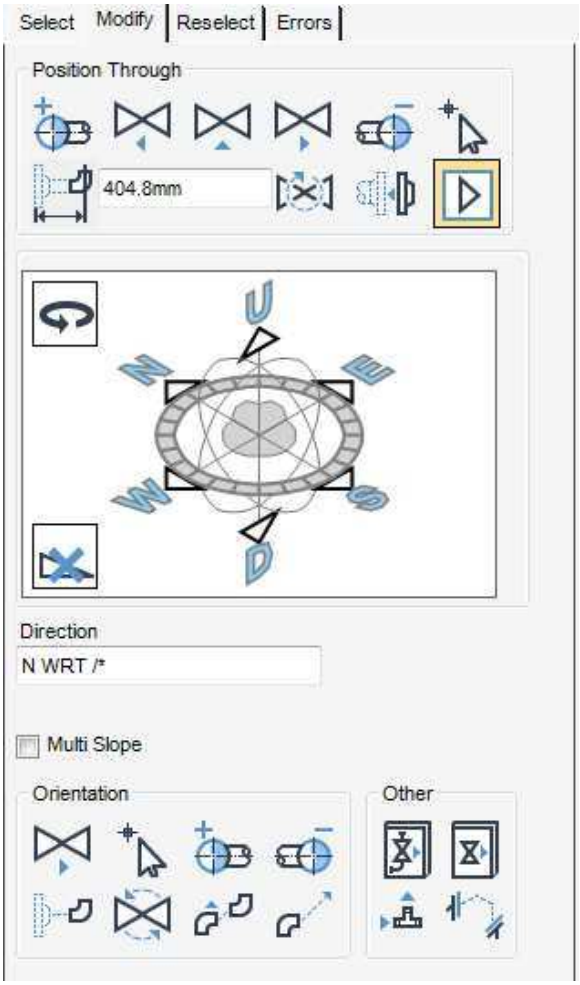


Clicking the pipe name to modify the pipe attribute. The **Pipe Editor** form will be shown, we can modify name, specification, insulation,… see **1.2.1** for more detail.

The table **Connectivity** show the head and tail details and connection information. The selected branch will also be highlighted in the **3D View**

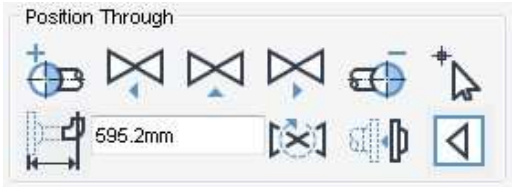


4.2.2.2 Modifying Piping Components

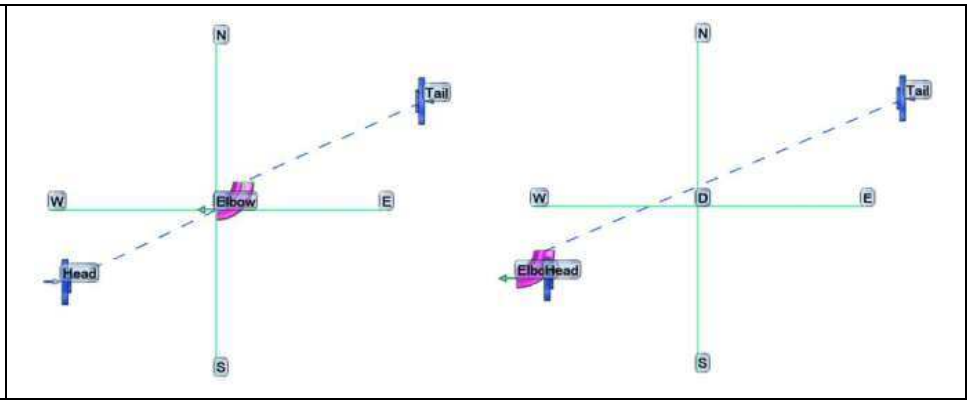


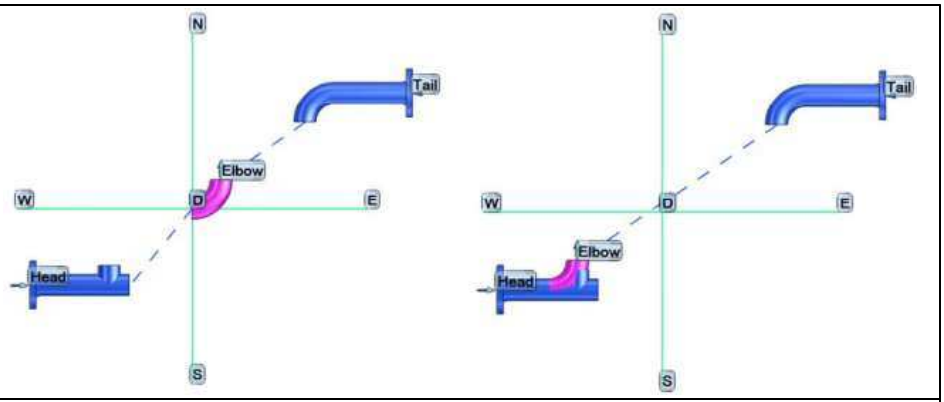
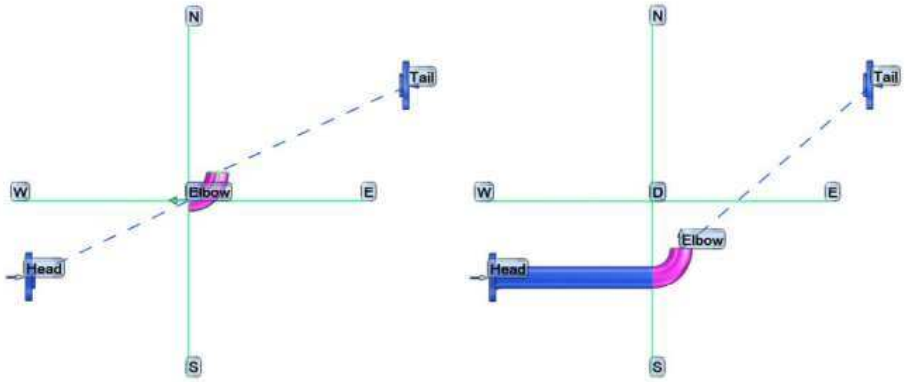
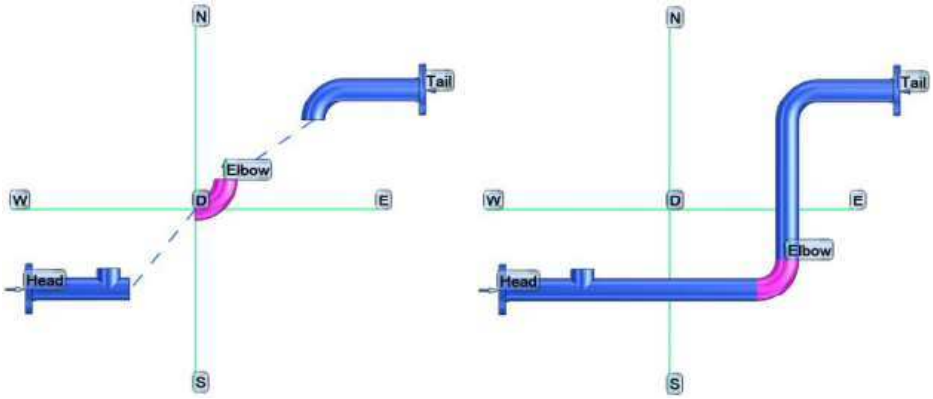
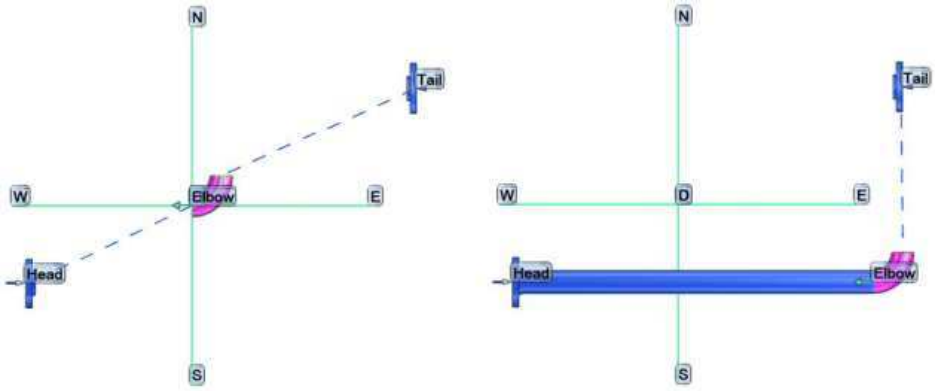
Piping components, their orientation and position can be modified.

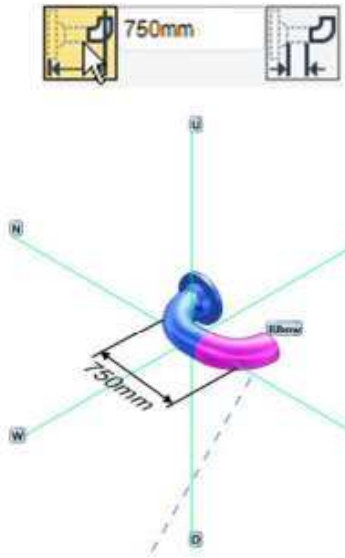
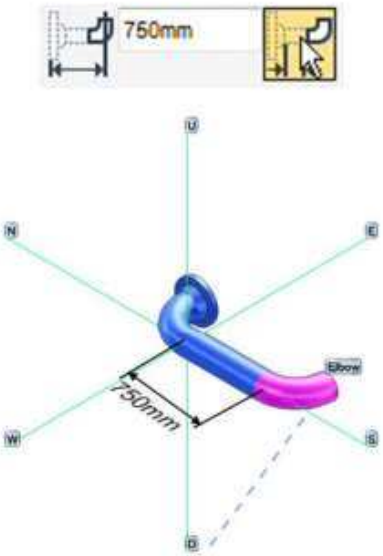
Position Through.



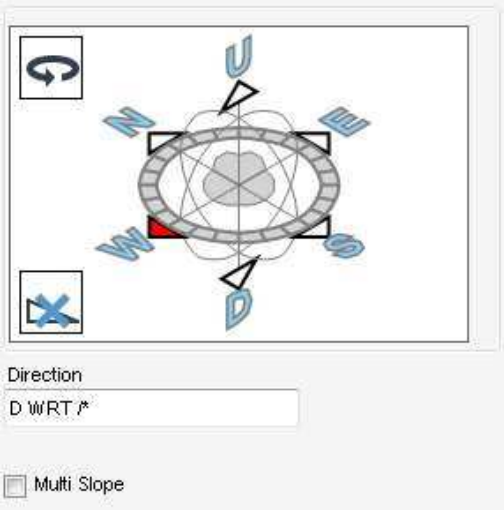
Use the **Position Through** part of the **Piping Component Editor** window to adjust the position of a component.

	Position through branch head	

	**Position through previous element**	
	**Position align with previous element**	
	**Position through next component**	
	**Position through branch tail**	
	**Position through a cursor pick**	Displays the **Positioning control** window.

	**Distance – click to use Clearance**	
	**Distance – Click to use Distance**	
	**Copy current component**	Copies the currently selected component.
	**Change to backwards mode**	Change the routing direction form tail to head.
	**Change to forwards mode**	Changes the routing direction from head to tail.

Rotation & Direction.



The **3D Compass** part of the **Piping Component Editor** window can be used in two ways depending upon the type of component selected. By default, the **Direction Tool** is displayed as default.

Direction Tool: the direction of elbows and bends can be changed.

Rotation Tool: tees and other non-symmetrical components such as valves or reducers can be rotated.

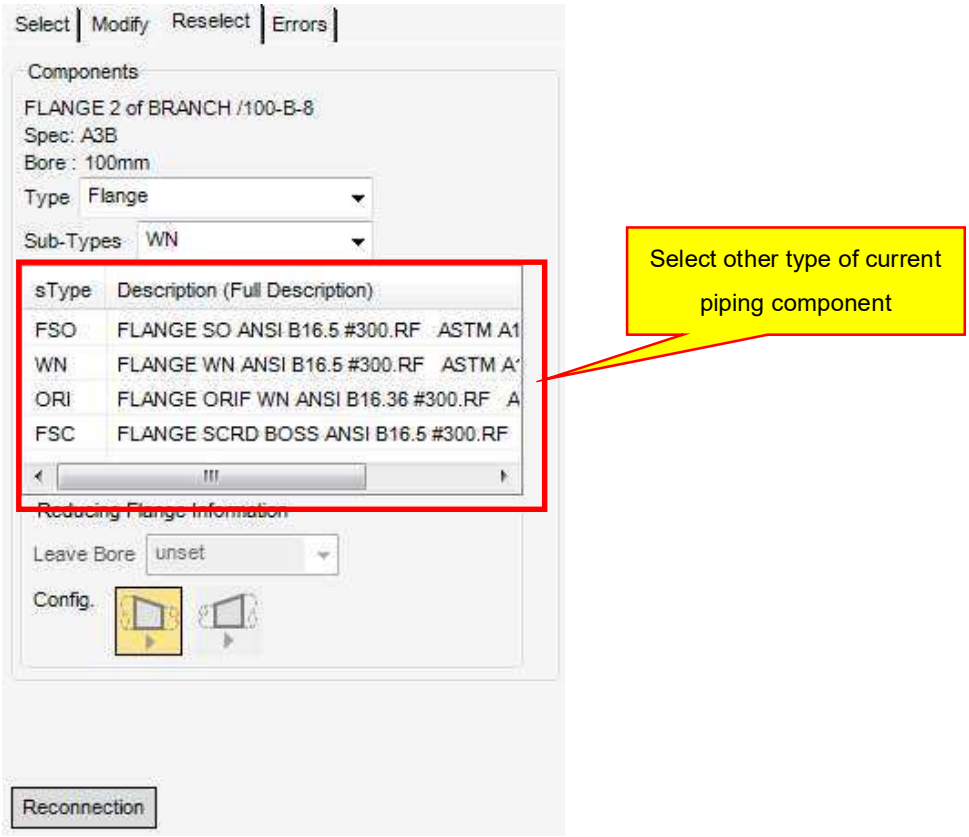
Use **Component Orientation** to adjust the orientation of a component.

	**Direction towards next**	Sets the orientation of a component towards the next component in the branch assembly (Corresponding command : dir to next)
	**Direction towards previous**	Sets the orientation of a component towards the previous component of the branch assembly. (corresponding command : dir to pre)
	**Direction towards pick**	Picks a position in the 3D graphical view to direct a component towards
	**Direct towards head**	Sets the orientation of a component towards the head of the branch assembly
	**Direct towards tail**	Sets the orientation of a component towards the tail of the branch assembly.
	**Orientate current component**	Sets the orientation of the arrive point of the current component to the same orientation as the leave point of the previous component.
	**Swap arrive and leave points(flip)**	Flips a component.
	**Align/selection component**	Aligns the selected component with the arrive and leave direction of the branch leg.
	**Direct selection / component**	Aligns the selected component to the next/previous component within the route, where possible.

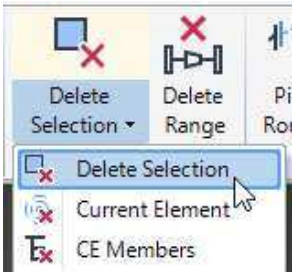
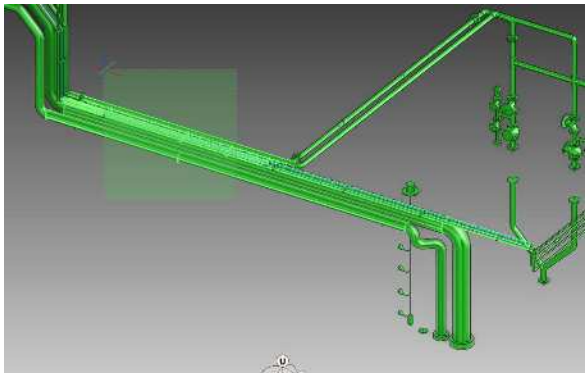
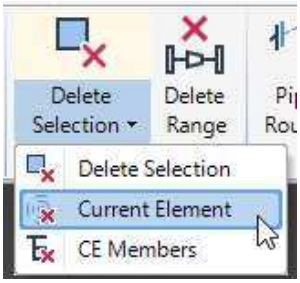
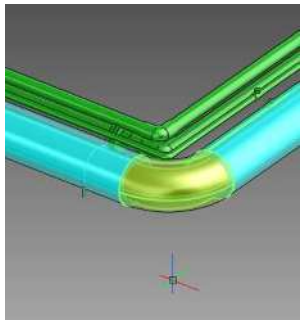
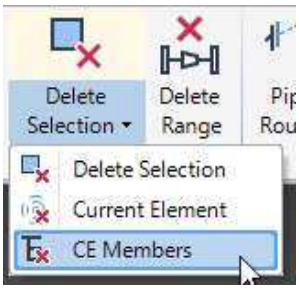
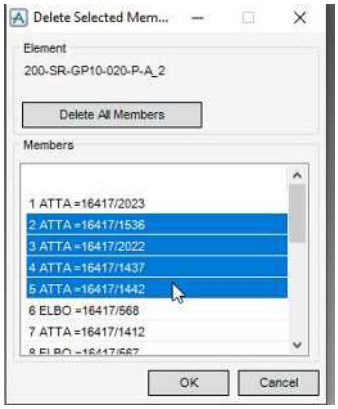
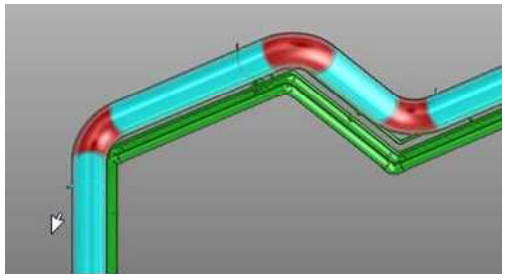
	Advanced positioning Drag	Displays the Drag Move window when a component is selected in the Model Explorer.
	Advanced positioning Move	Displays the Move window when a component is selected in the model explorer. Displays the move branch window when a branch is selected in the Model explorer.

Reselect Component.

Use the **Reselect** tab to replace a piping component from a list of available components, select the **Reselect** tab to display the CE's attributes and properties.



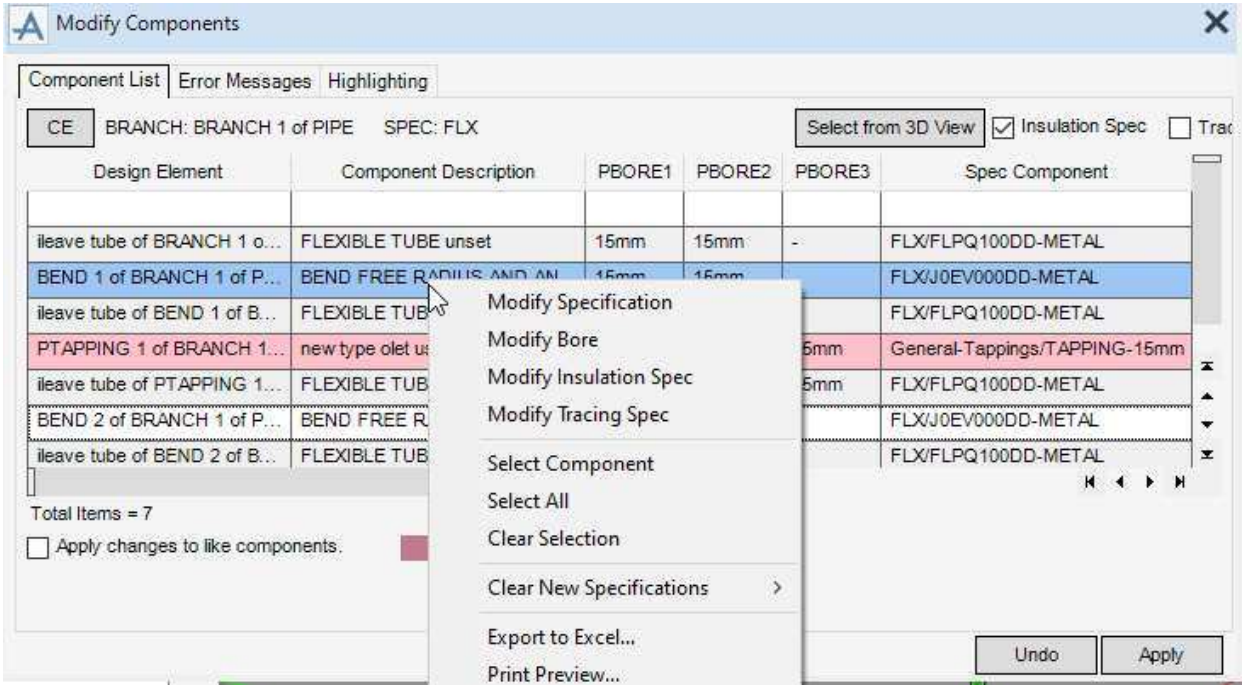
4.2.3 Delete Group

Function	Description	Image
	Select the object that you want to delete then click to Delete Selection. You can drag the mouse from left to right or right to left to select the large number of object (as the picture in the right)	
	Click to the Current Element to delete the current element on 3D view. (the current element normally displayed on 3d view in yellow colour).	
	Click to CE Members to delete member of a branch or a pipe. (You can delete one member, more than one member, or all members)	
	You can delete a range of members by clicking to “Delete Range” bottom as the picture in the left. (for example: you can delete a range of members in a branch by Identifying the start and the end of range of the selection that you want to delete in 3D view.	

4.2.4 Tools Group

4.2.4.1 Pipe Component Bore and Specification

A pipe or branch and the components contained within can be viewed and their attributes modified. The same window is displayed for the currently selected.



There are some modifications we can use:

- Modify Specification: modify specification of selected items.

- Modify Bore: modify bore of selected items

- Modify Insulation Spec: modify insulation spec of selected items.

- Modify Tracing Spec: modify tracing spec of selected items.

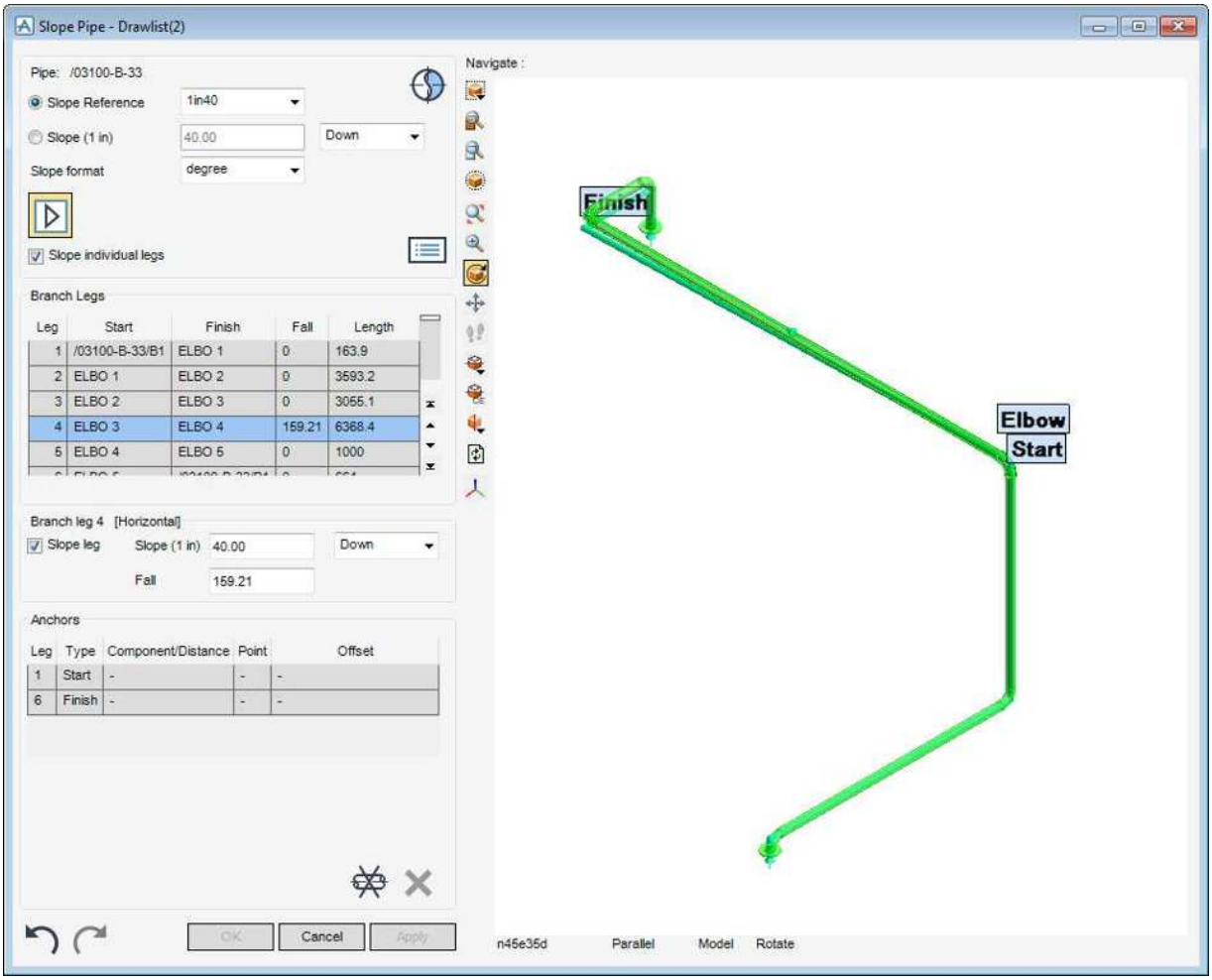
- Select Component: reselect piping components

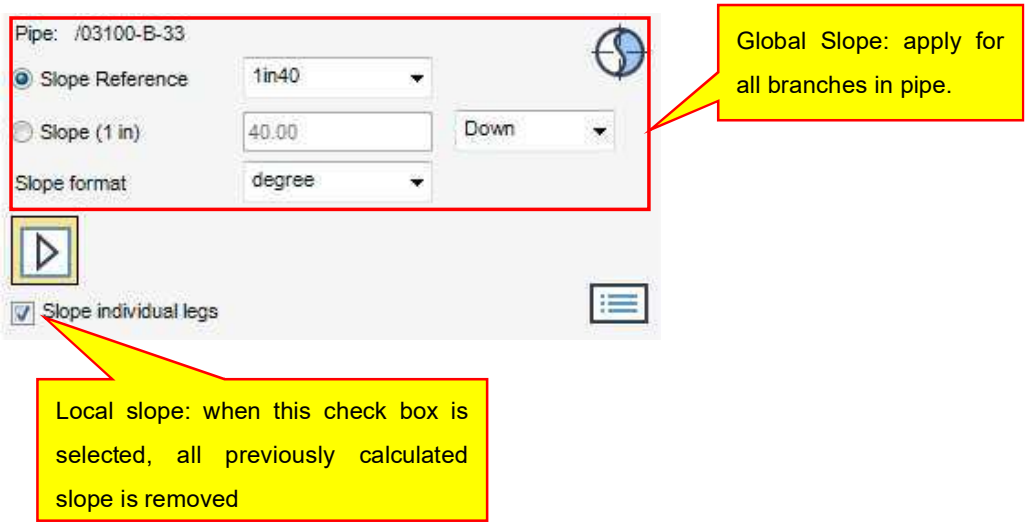
- Select All: select all items in table.

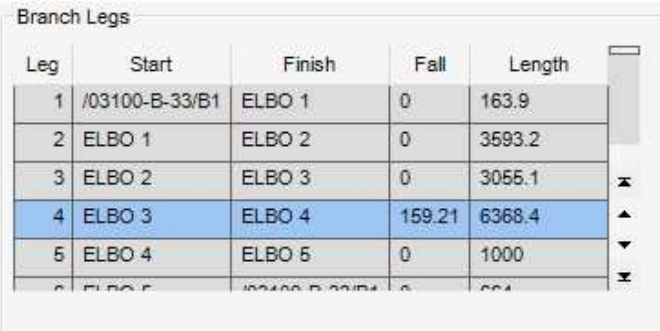
- Clear Selection: clear current selection.

4.2.4.2 Pipe Slope

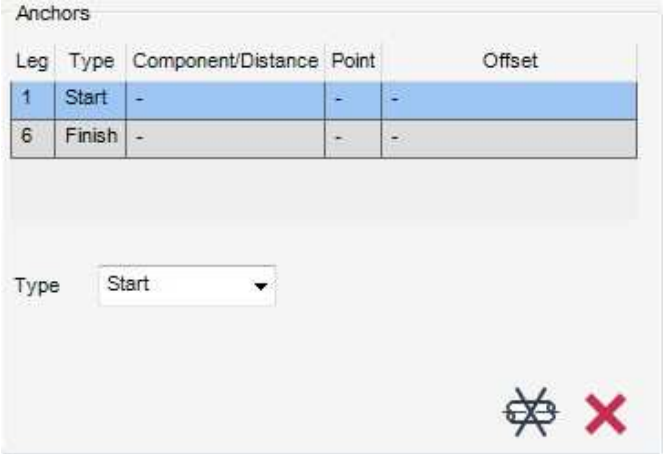
Use the dual purpose window **Slope Pipe** window to view the details of the CE and if required define and apply a slope to any number of branches contained within the pipe element.





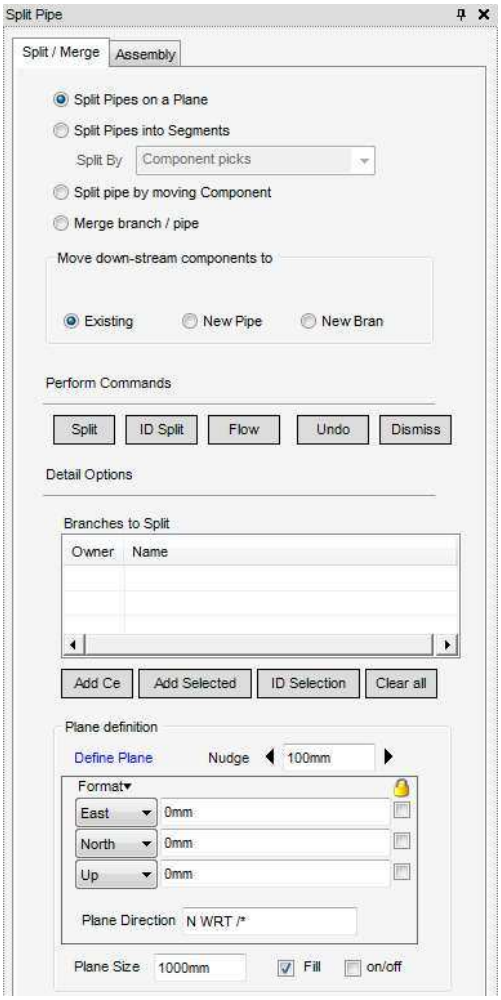


If we want to set the slope for every branch leg, select every branch leg in **Branch Legs** pane and change the information of slope in the **Slope Branch Leg** pane.

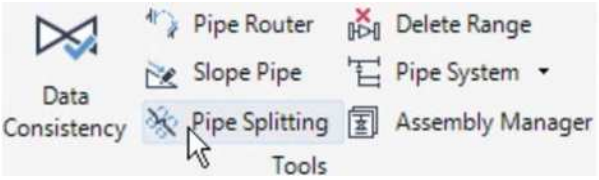


If necessary the selected branch leg can also be anchored to two different locations by, adjusting the vertical position of the leg to pass through the anchored positions. One or two anchor positions can be used.

4.2.4.3 Pipe Splitting



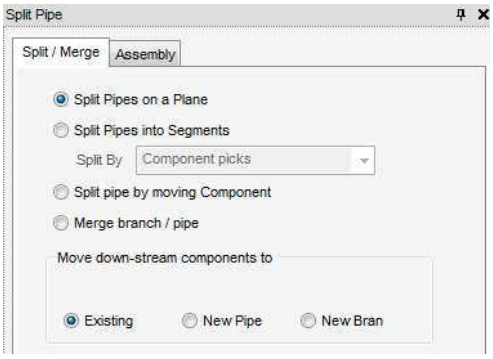
All of the tasks that are associated with the splitting of pipes is initiated from a central **Split Pipe** window which acts as a task hub. By default, the **Split/Merge** tab is selected.



The **Split Pipe** form has two tabs:

- **Split/Merge** – this is the main tab used to set up the **split** and **merge** functions.
- **Assembly** – this is used to select the assembly that will be inserted when splitting.

Splitting Options



There are three radio button options for splitting:

- **Split Pipes on a Plane**: splits the pipe by inserting an assembly component at the intersection with a plane.
- **Split Pipes into Segments**: splits the pipe into segments of a specific length. The splitting is defined between two points that are indicated in the 3D view.
- **Split Pipes by Moving Component**: splits the pipe at the selected component. The selected component and all those downstream will then be added to a new pipe or branch.

Move down-stream components to section:

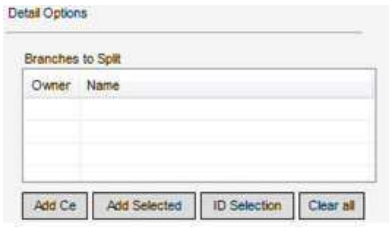
- **Existing**: components are inserted at the split position in the currently selected branch.
- **New Pipe**: components downstream of the split are inserted into a new pipe in the hierarchy.
- **New Branch**: components downstream of the split are inserted into a new branch owned by the current pipe.

Split Pipes on a Plane.

The regular splitting pipe procedure:

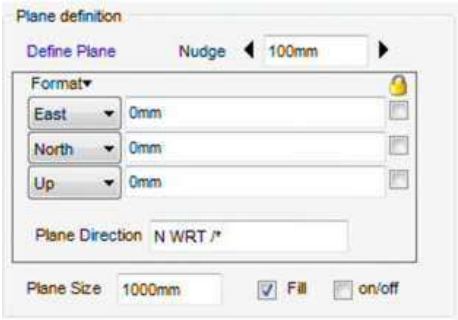
- Select branches to split
- Define the plane is used for splitting.
- Perform the command.

Select branches



Add Ce	Add the branches related to the CE to the list
Add selected	A number of pipes can be selected in the 3D view by dragging a crossing window across the required pipes with the cursor. The selected pipes are then added to the list by clicking the **Add Selected** button.
ID selection	Similar to above. This list can be dynamically updated by picking any pipe component. When the selection is complete, the escape key is pressed to end selection
Clear All	To clear the **Branches to Split** list.

Define splitting plane



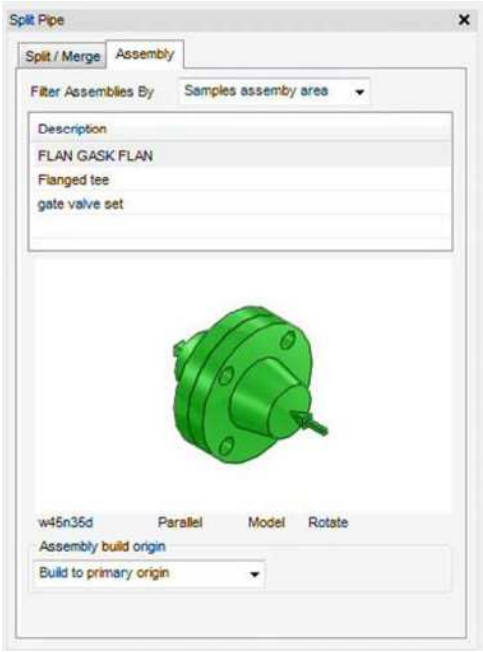
Click **Define Plane** button to perform the selection of a position in a 3D view and set it as the cutting plane.

After the selection is complete, the position of splitting plane will be shown in the Position pane. Change the position directly by change the number in every input box.

Perform the command

Split	Perform the splitting work for all selected branches
ID Split	proceed the splitting work by selecting one by one branch (which have added to **Branches to split**)
Flow	Add a flow direction arrow to the 3D View to indicate which side of the split represents the downstream components
Undo	Back to previous status
Dismiss	Closes the form

Split Pipe into Segments.



This function is used to split pipe into segments of a given length between selected components or selected features that are adjacent to the pipe. This function cannot be used for multiple branches at the same time.

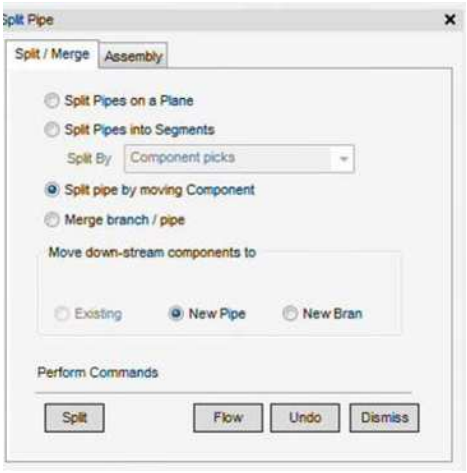
The **Split Pipe Length** section of the form determines how lengths are to be used by the function. Select from the **Segment Length** or **Cut-pipe Length** options and key in the required value.

- **Segment Length**: split length is always included the length of component.
- **Cut-pipe Length**: split length is tube length.

The **Minimum Final Tube Length** restricts the length of the final tube. If the final tube length is below this value, the previous spools are adjusted to make the final length within this value.

Split by Moving Component.

This function allows a single branch to be split at an indicated component. It does not require the creation of a plane, and is not capable of handling multiple branches. The indicated component will be moved to the separated object.

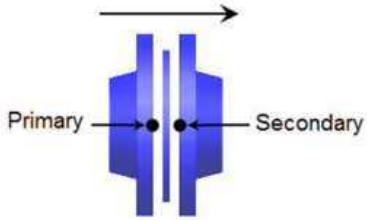


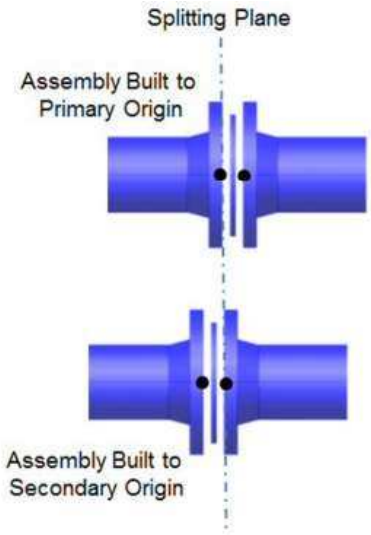
Choose the option **Split pipe by moving Component**, then choose the Move down-stream components to section. And Click the button **Split** to proceed the picking a piping component mode. After the selection, pipe will be splitted.

Assembly Tab.

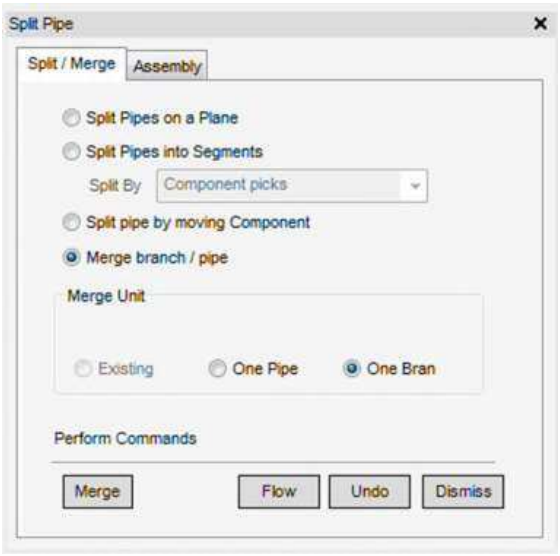
When using **Split Pipes on a Plane** and **Split Pipes into Segments** options, user need to selected type of assembly items which will be inserted into split position.

The **Assembly build Origin** options list determines whether to build the assembly using the primary or secondary origin, as defined during the assembly creation. Example about the primary and secondary origin points





4.2.4.4 Merge Pipe/Branch



The **Merge branch / pipe** function provides the following functionality:

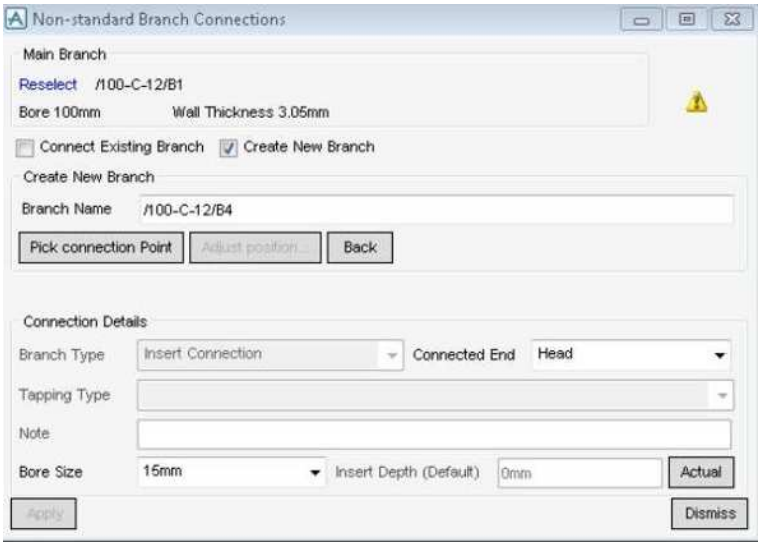
- Merge two adjacent branches
- Merge two adjacent pipes.

4.2.4.5 Non Standard Branch Connections

A non-standard connection is essentially a branch connection at any point on the surface of the main branch. Normally these are tube connections which are inserted by drilling a hole at some point and welding the new branch into place.

With non-standard branch connections you can create and connect new branches or connect existing branches to components such as valves, reducers and elbows. The branch connects to a Cartesian point (PIPCA) owned by the component or an element called PTAP for tube to tube connections.

Unlike other TEE or OLET type connections a special branch connection does not require additional components in the piping specification but it does require specification /General-Tappings.



Click **Select** button in **Main Branch** pane to choose the main branch for taking branch connection.

There are two options for connections :

- **Connect Existing Branch**: connect an existing branch to main branch.
- **Create New Branch**: create a new branch and connect it to main branch.

Continue the process by click button **Pick connection Point** to select a position on the tube surface on main branch to identify where to put the branch connection. After selection, user can modify the selected position by button **Adjust Position**.

The **Branch Type** can be selected form the **Connection Details** pane:

- Insert Connection: The branch tube is inserted into the main branch and welded.
- Surface Connection: The branch tube is placed on the main branch and welded.
- Saddle Connection: The branch tube is shaped to fit to main branch and welded.
- Extruded Connection: The main branch tube is shaped by machine to form a tube to tube connection.
- Boss Connection: a fitting is selected from the spec and it is welded to form the branch connection (Boss type elements must be in the branch specification, have a Gtype of COUP and COMPTYPE eq 'TAPP' to be recognized by this application)

The **Insert Depth** value :

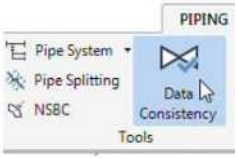
- Default: The standard calculation for this type of branch.
- Actual: the actual insert depth on the current connection.
- User: The current value entered but not yet fixed by clicking Apply. The button to the right of the insert depth will swap the insert values between actual and default.

4.2.4.6 Data Consistency Check

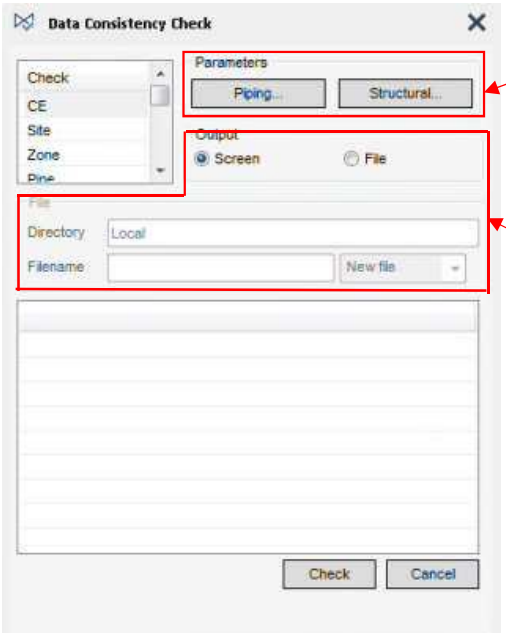
The data consistency checking utility, available within the Piping application, checks the design for occurrences of the following types of error:

- Angular alignment
- Axial alignment
- Consistent bores
- Connection types
- Minimum tube length

Accessing the data consistency checks



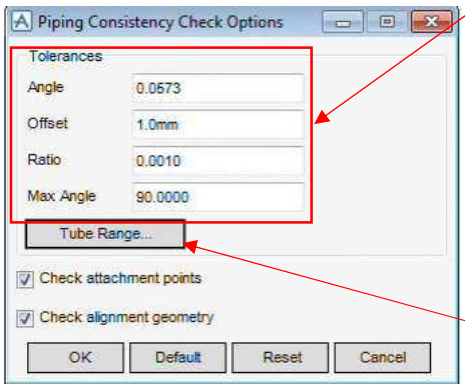
In the **Tools** group click the Data Consistency button to show the Data Consistency Check form as shown



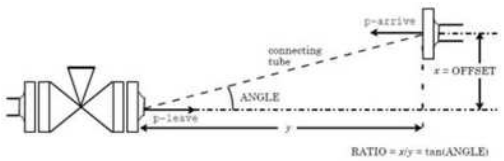
Parameters: check allows a margin of acceptable error before a problem is reported. There are 2 objects for checking: piping or structural

Output: send the report to **Screen (PDMS's command line)** or **File**

Piping Consistency Check Options



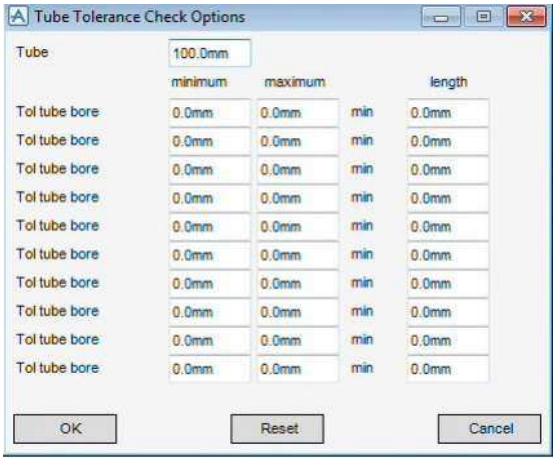
Tolerances: margin of acceptable error. Picture below will explain the meaning those terms in Tolerances



The maximum permissible values for any of these parameters can be specified

Tube Range: click this button to enter the **minimum Tube Length options**

Minimum Tube Lengths



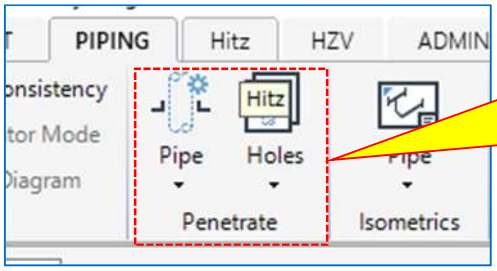
By default a report will warn of all lengths of tube in the design which are shorter than 100mm. This allows a decision to be made whether each such length is adequate for welding procedures, bolt withdrawal, access, and so on. This minimum length can be changed from the default of 100mm, and may set different minima for up to ten different pipe bore ranges if desired.

Data Consistency Diagnostic Messages

Full list of the data consistency diagnostic messages, each identified by a reference number can be found in the Model Reference Manual of the VAEVA Everthing3D help.

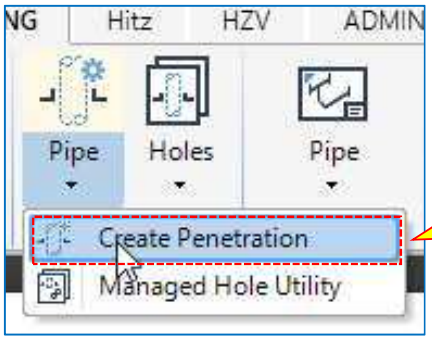
4.2.5 Penetrate Group

Penetrate Group Interface.

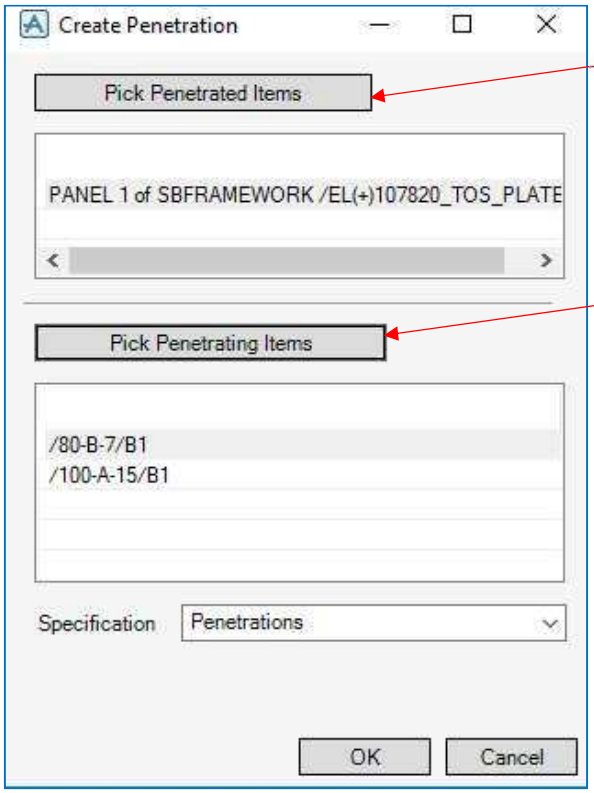


This group allow you to create, modify and manager the Pipe Penetration and Holes. Their functions will be introduce follow Aveva manual.

Create Penetration.



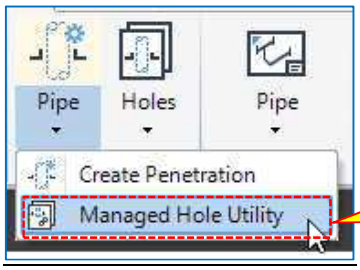
To request a Piping Penetration, in the Penetrate group, locate and click Create Penetration to display the Create Penetration window



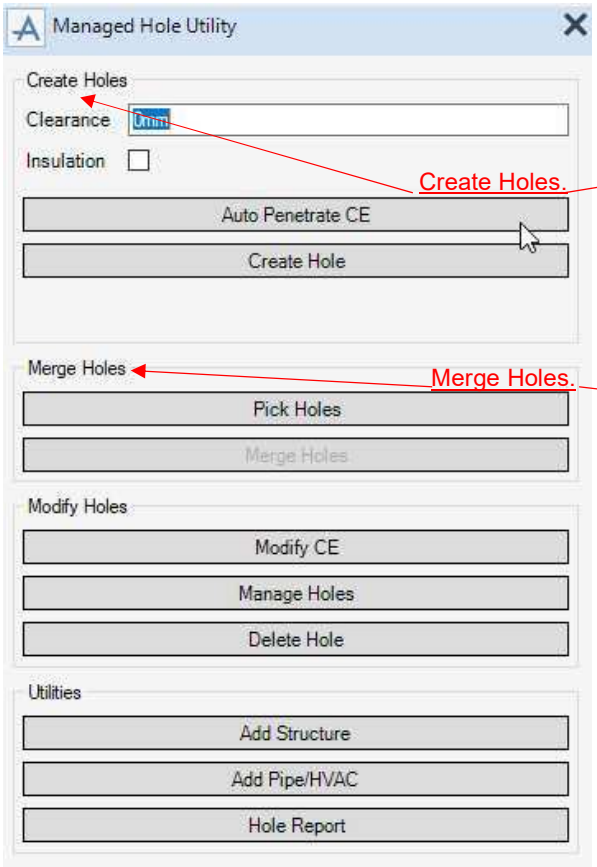
Pick to select Penetrated Items.

Pick to select Penetrating Items.

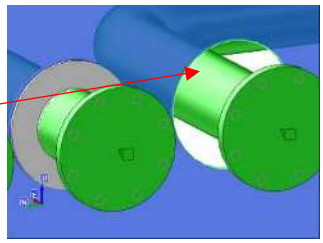
Managed Hole Utility.



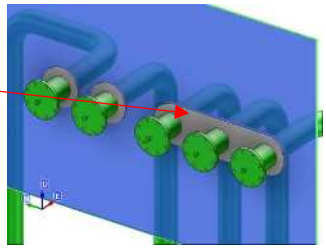
In the Penetrate group, locate and click Managed Hole Utility to display the Managed Hole Utility window



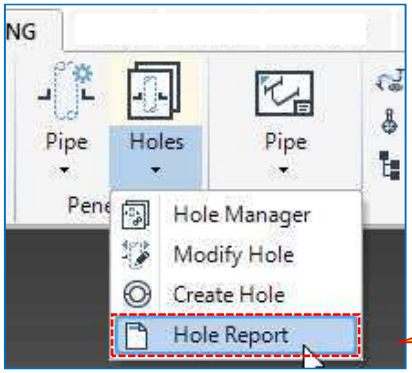
Create Holes.



Merge Holes.



Managed Hole Utility.

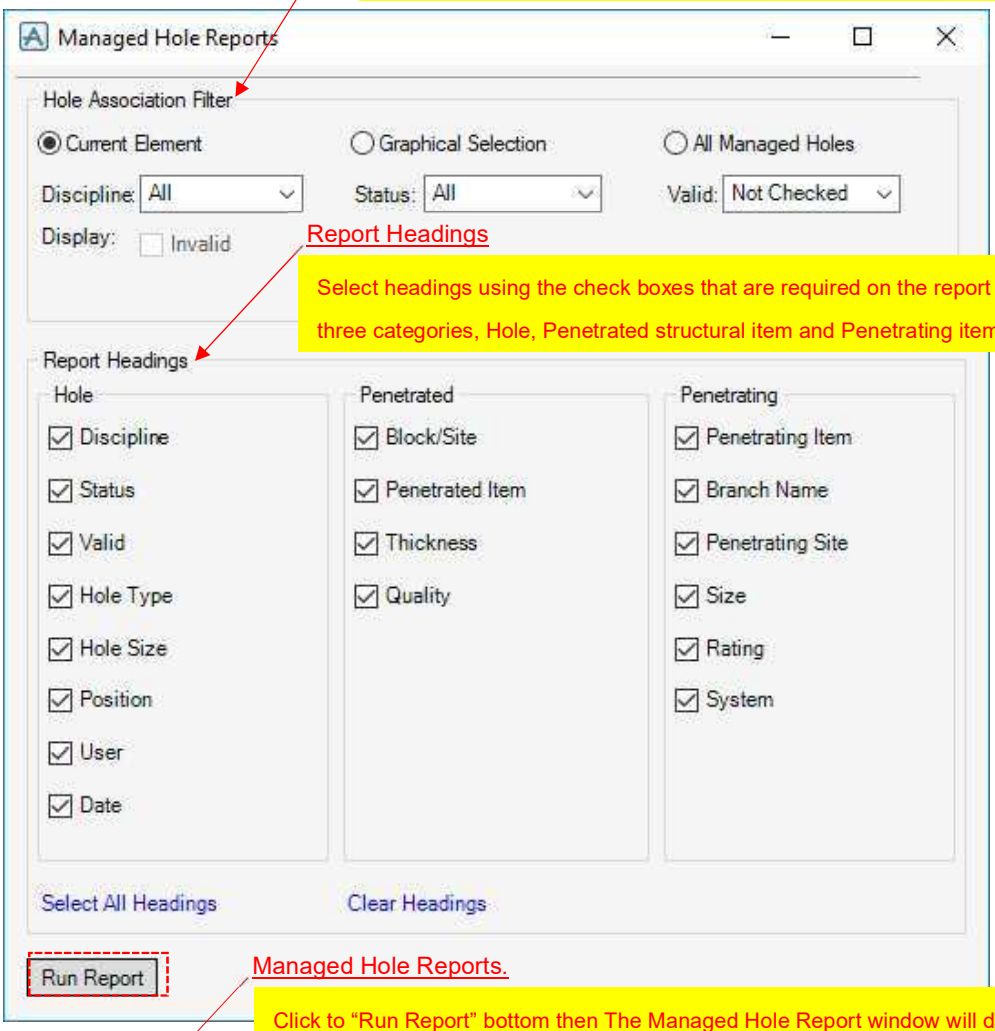


Click Hole Report to open Managed Hole Reports window.

The **Hole Association Filter** pane searches for Managed Holes and displays the search results in the Hole Report. The filter criteria specify which Managed Holes is selected:

Hole Association Filter

Searches for Managed Holes and displays the search results in the Hole Report. The filter criteria specify which Managed Holes is selected

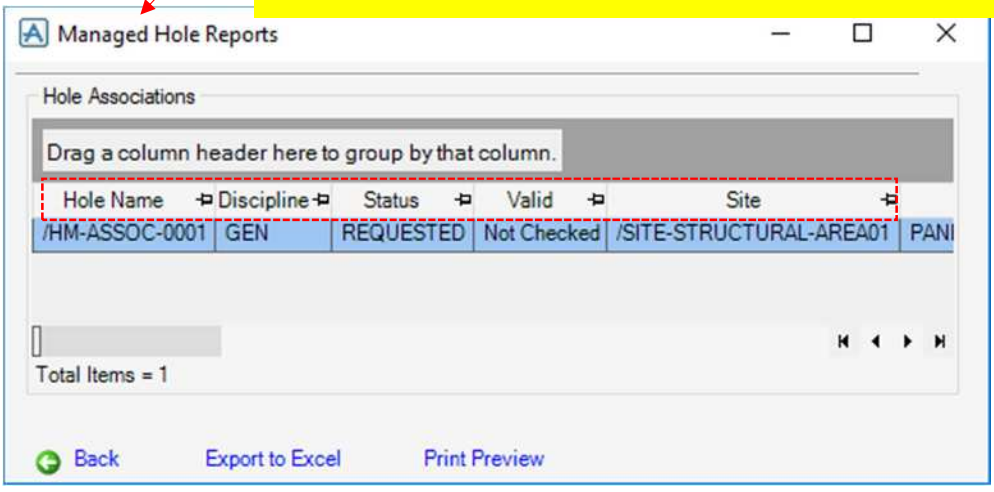


Report Headings

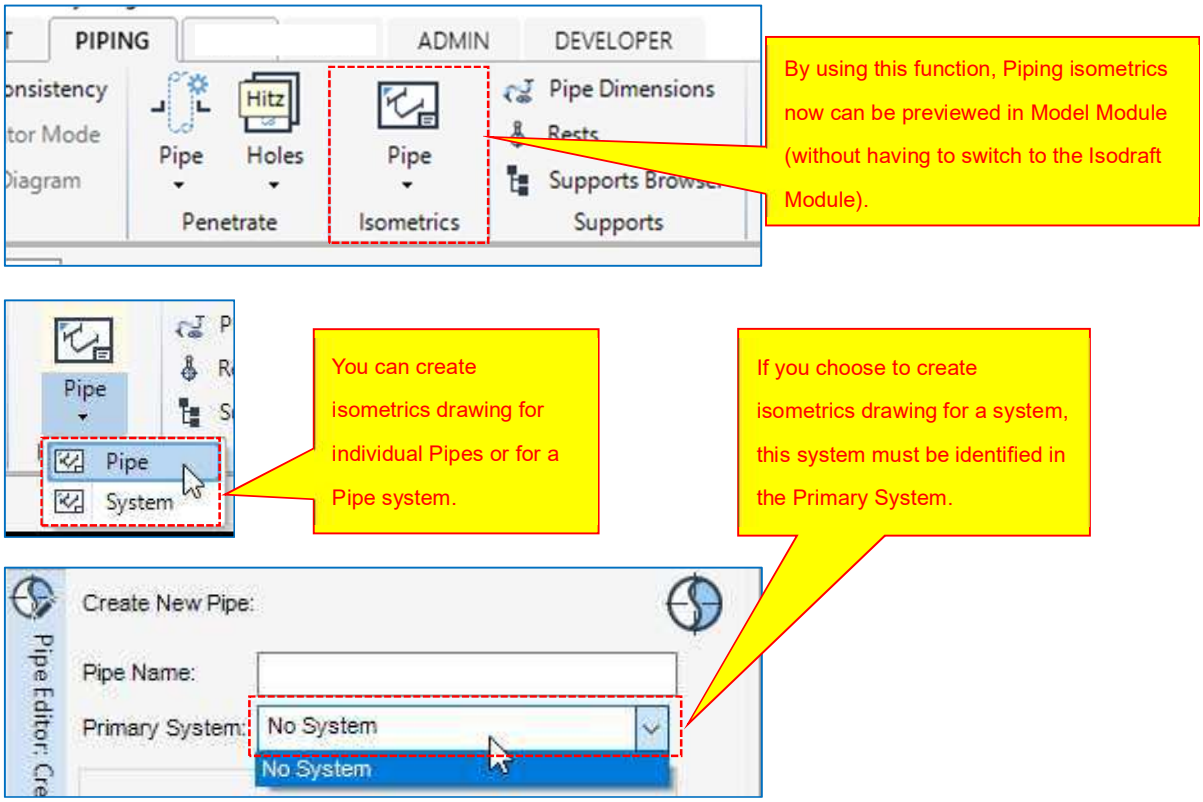
Select headings using the check boxes that are required on the report from the three categories, Hole, Penetrated structural item and Penetrating items.

Managed Hole Reports.

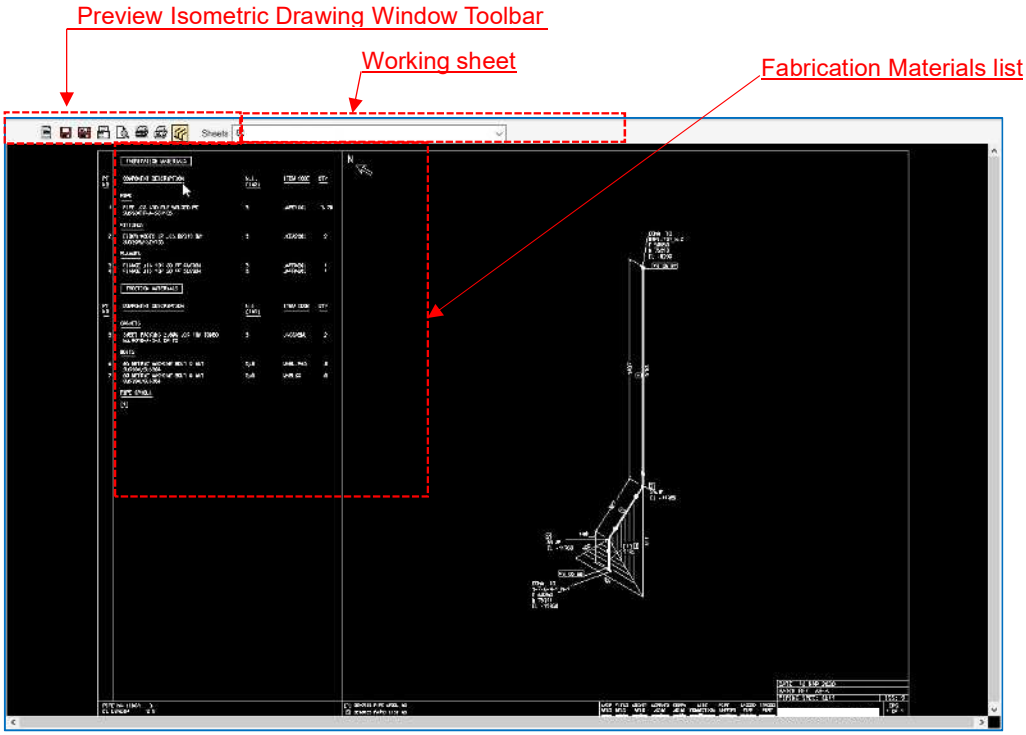
Click to “Run Report” bottom then The Managed Hole Report window will display the report generated by the specified selection criteria and report headings.

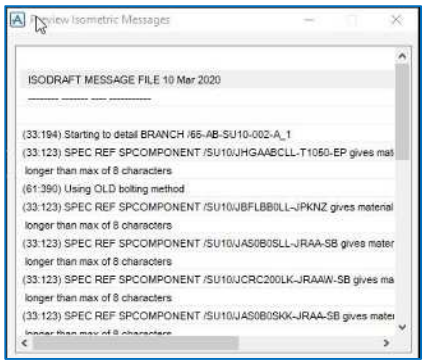


4.2.6 Isometrics Group

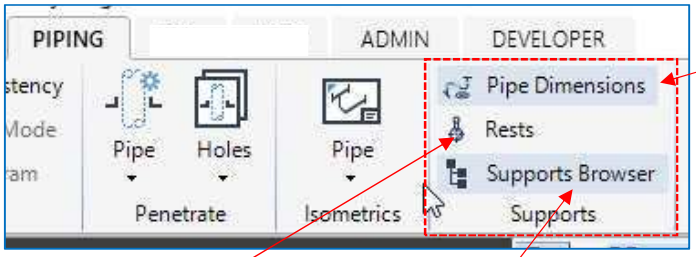


The **Preview Isometric Drawing** window is displayed as below.



Function	Description	Image
View the message file	Displays the **Preview Isometric Messages** window, where messages created during the generation of the isometric drawing can be viewed.	
Save As	Displays a **Browser** window, save the preview isometric drawing as a plot file.	
Save All	Displays a **Browser** window, save all the sheets of the preview isometric drawing as plot files.	
Print Setup	Displays the printer settings.	
Print Preview	Displays an on screen preview of the isometric.	
Print	Prints the currently selected preview isometric drawing.	
Print All	Prints all the sheets of the preview isometric drawing.	
Highlight On/Off	Highlights all the display elements of the currently previewed isometric drawing sheet in the 3D View. Click again removes the highlighting. If the elements are not currently displayed in the 3D View, click twice. The first click adds the elements to the 3D view, the second highlights the elements. Additional clicks turn the highlighting on and off. N.B. The highlighting only occurs if the Isometric Options file uses the Hitfile option.	

4.2.7 Supports Group



Pipe Dimensions

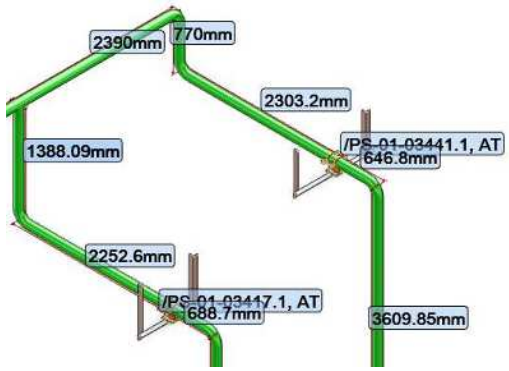
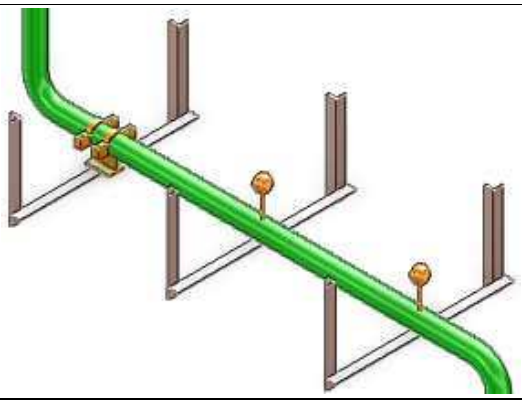
Displays dimension tags for pipes and branches in the 3D view

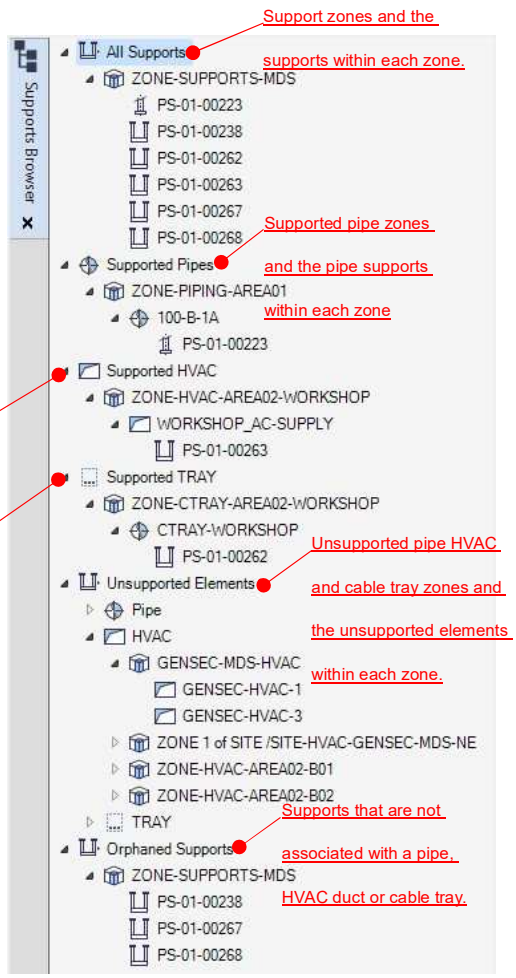
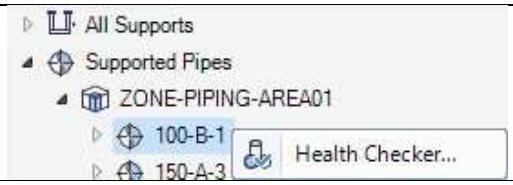
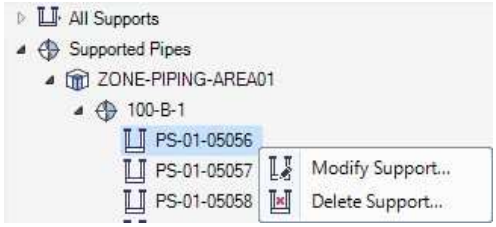
Rests

Displays pipe rests in the 3D view

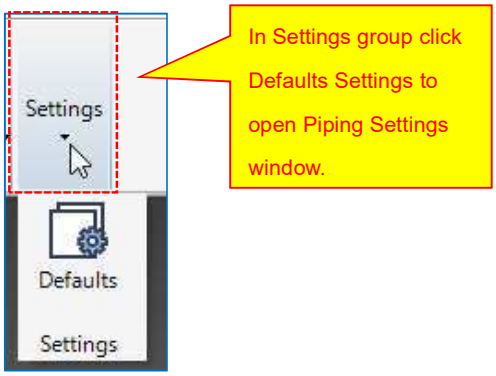
Supports Browser

Provides a graphical method for viewing different support types within the project

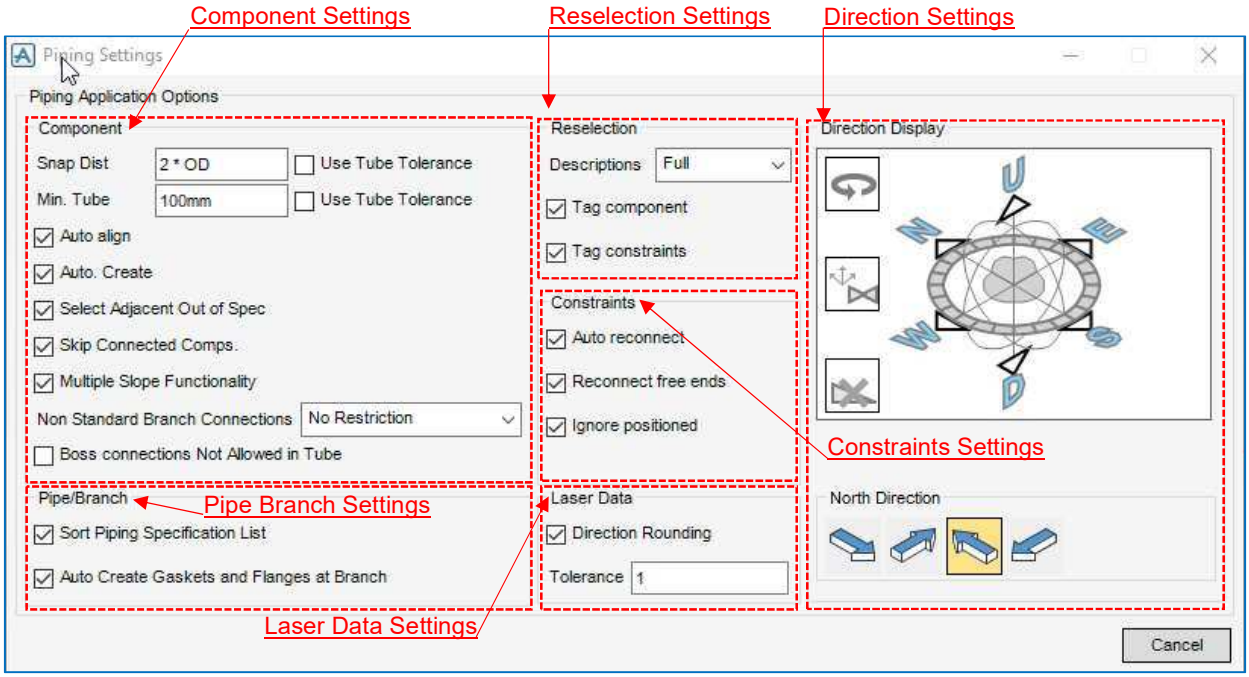
Function	Description	Image
Pipe Dimensions	- Displays dimension tags for pipes and branches in the 3D view. The functionality also displays the position and function type of supports on the pipe. - Click to identify the branch using a cursor pick in the 3D view. You can continue to identify additional branches on the pipe. - Press ESC to end the selection process and display the dimension tags for the selected branches. - Repeat the command to remove the dimension tags from the 3D view.	
Rests	- Displays pipe rests in the 3D view. - All pipe rests are highlighted in the 3D view with a pin. - Repeat the command to remove the pin from the pipe rests in the 3D view.	

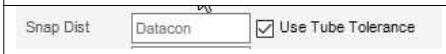
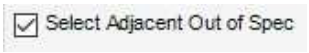
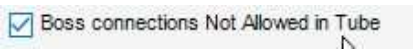
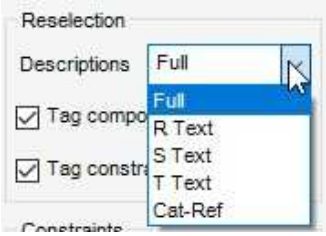
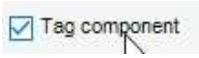
Supports Browser	- Provides a graphical method for viewing different support types within the project. - The Supports Browser displays the support elements in a hierarchy. To expand an element hierarchy, click to the left of the element name. To contract an expanded element, click to the left of the element name. Supported HVAC zones and the HVAC supports within each zone. Supported cable tray zones and the cable tray supports within each zone.	
Health Checker...	Right-click a supported ZONE element to Display a drop-down menu.	
Modify Support...	To modify a support, right-click a SUPPORT element to Display a drop-down menu. You could also delete this support from the drop-down menu.	

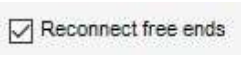
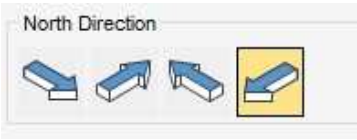
4.2.8 Settings Group



The **Piping Settings** window is displayed as below:



Function	Description
	Sets the default snap distance (the minimum distance between the end of a component and the placement of a new component between which an automatic connection occurs) when creating a component. The selection is connected to the nearest component if placed within the set snap distance. The default is 2 * OD.
	Uses a pre-set Data Consistency specific tube length to ensure the pipe is placed correctly.
	Sets the minimum tube distance between two components, the default is 100mm.
	Uses pre-set Data Consistency specific tube length to ensure the pipe is placed correctly.
	Newly created components are aligned to the previous component automatically.
	The software determines if there is a requirement for associated components by default (for example, the associated components for a valve are a gasket and a flange).
	An alternative specification component can be used.
	The piping application determines which components are connected, then positions the creation point approximately.
	Displays the point direction option on the **Piping Component Editor** window.
	Sets the defaults for the non-standard branch connections.
	For pipe tappings, specifies the boss type connections (for example, olets, half couplings, welded bosses) that can only be used on fittings and excluded from the tube.
	Sorts the piping specification list alphabetically.
	On creation of a component, gaskets and flanges are automatically created.
	Sets the display of the Description in the Component Type List from the Select and Reselect tabs.
	Sets the display value for the component tag in the 3D graphical view.

	When a component is selected, adjacent constrained components are highlighted in the 3D graphical view.
	Reconnects components that are modified using the **Reselect** tab. Reconnection errors display in the Errors tab.
	Reconnects the branch head or branch tail if reselection of a component results in disconnection.
	Ignores positioned components. Positioned components will normally not be moved when reconnected.
	Tolerance of the direction. Rounds to an orthogonal direction (ENUWSD) for each component from the laser pick.
	When you choose the North Direction, the The **Direction Display** part of the Piping Application Defaults window sets the default display settings on the Piping Component Editor window.